62nd Annual Meeting of the APS Division of Fluid Dynamics

Minneapolis, Minnesota

http://www.aps.org/meetings/meeting.cfm?name=DFD09

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AA Turbulent Boundary Layers: Experiments I 101A

8:00AM AA.00001 ICET - International Collaboration on Experiments in Turbulence: Coordinated Measurements in High Reynolds Number Turbulent Boundary Layers from Three Wind Tunnels H. NAGIB, A. SMITS, I. MARUSIC, P.H. ALFREDSSON, AND ICET TEAM — Zero pressure gradient (ZPG) boundary layers are one of the canonical, wall-bounded, turbulent flows that have been the focus of experimental and analytical investigations for several decades. Over the past few years, four groups have focused on systematic comparison between several measurement techniques and three facilities. Two closed return wind tunnels with ZPG boundary layers developed on a plate suspended near the mid-height of the test section (at KTH and IIT), and an open return facility with a large and long test section and a boundary layer developing along its floor (at the University of Melbourne), are used for these coordinated efforts. The development length of the boundary layers and the free-stream velocity in the three facilities range from 5.5 to 22 m, and from 10 to 60 m/s, respectively. Various arrangements for adjustable test section ceilings are employed to generate ZPG boundary layers over the range of momentum thickness Reynolds numbers from 11,000 to 70,000. Oil film interferometry (OFI) is employed to directly measure the wall shear stress, and various sizes of Pitot probes and types of hot-wire sensors are used to measure wall-normal velocity profiles at different locations and free-stream velocities. Mean velocity, turbulence statistics and integral parameters are examined.

8:13AM AA.00002 Accurate and Independent Measurements of Wall-Shear Stress in Turbulent Flows , J.-D. RÜEDI, R. DUNCAN, S. IMAYAMA, K. CHAUHAN, AND ICET TEAM — Oil Film Interferometry (OFI) is used to directly measure the wall-shear stress in the high Reynolds number turbulent boundary layers from three facilities used for ICET. Various optical arrangements were utilized to collect the digital images generated on transparent plugs integrated into the boundary layer surface. Test-section free stream velocities ranging from 10 to 60 m/s and development lengths from 5.5m to 21 m, resulted in friction velocities varying from 0.35 to 1.65 m/s, corresponding to boundary layer thicknesses varying by factors of nearly four. Silicon oils with viscosities from 20 to 1000 cSt were employed in the measurements, with multiple oils used for several of the test conditions. A reference temperature measurement was used in all three facilities and for the calibration of the oils as a function of temperature in four different laboratories using two types of viscometers. The processing of the images was carried out using several approaches and compared for consistency of the results. Results of the skin friction coefficient from the three wind tunnels are examined and compared as a function of the displacement thickness Reynolds number, as determined from hot-wire and Pitot probe profiles at comparable conditions, and are found to be accurately represented by the logarithmic Rotta relation. The various uncertainties and the final accuracy of this type of measurement are discussed.

8:26AM AA.00003 Pitot probe corrections for measurements in turbulent boundary layers , JASON MONTY, SEAN BAILEY, MARCUS HULTMARK, BEVERLEY MCKEON, AND ICET TEAM — Mean velocity measurements over a range of Reynolds number have been taken in zero-pressure-gradient, flat plate turbulent boundary layers using Pitot probes of varied diameter. Three world-class boundary layer facilities were involved in this investigation, ensuring the results are not facility-dependent. Different methods of correcting Pitot probe data were compared to each other and to a concurrent study where hot-wire measurements provided mean velocity data. It was found that there was very little difference in the commonly used shear corrections, although improvements could be made in the near-wall corrections and a modification to the correction is proposed. The applicability of a turbulence correction is investigated with the final, fully corrected pitot probe measurements compared with hot-wire measurements, demonstrating excellent agreement overall between the two. This study confirms the accuracy of pitot probes for mean flow analysis in turbulent shear flows.

8:39AM AA.00004 Challenges in Hot Wire Measurements in Wall-Bounded Turbulent Flows , R. OERLUE, N. HUTCHINS, T. KURIAN, A. TALAMELLI, AND ICET TEAM — Despite the rapid development of optical velocimetry methods (like LDV, PIV, etc.) the hot-wire anemometer remains the main instrument used in wind tunnel studies of turbulence. To obtain precise results close to walls in turbulent boundary layers, requires the user to have accurate procedures for a good calibration at low velocities, knowledge of effects of blockage and heat conduction to the wall, and how spatial resolution influences the results. We have carried out measurements in three different wind tunnels (at KTH, Univ. Melbourne and IIT) with various hot-wire probes (stubbed and stubless, as well as straight and boundary layer type) operated with commercially available and home-made anemometer systems. The use of different facilities enabled measurements at similar Reynolds numbers, but with different free stream velocities, resulting in a wide range of viscous scales for the hot-wire sensor lengths. The results indicate that poor spatial resolution influences the measured fluctuating velocity distribution well into the overlap region and clarifies controversial aspects regarding the scaling of the near-wall peak and the apparent existence of an outer peak in the rms distribution. The mean velocity within the buffer region has been found to be affected by probe geometry and size, an influence that is especially important when correcting for the absolute wall position by means of common correction methods.

8:52AM AA.00005 Mean Flow Measurements with Pitot Probes in High Reynolds Number Boundary Layers , S. BAILEY, M. HULTMARK, R. DUNCAN, B. MCKEON, AND ICET TEAM — Mean velocity profiles were measured in zero-pressure-gradient turbulent boundary layers using a variety of Pitot probes in three facilities at five different Reynolds numbers. The results were analyzed to verify the Reynolds number similarity between the two facilities and the scaling of the integral parameters was found to be consistent with the results from other studies. A linear relationship was found between Re_θ and Re_τ and used to determine a new friction factor relationship based on momentum thickness. Different methods used to determine the von Kármán constant were compared. When the friction velocity was determined using the new relationship, a logarithmic region between $y^+ = 300$ and $y/\delta = 0.1$ was observed for $Re_\theta > 15000$. A von Kármán constant of 0.40 was found for this region.

9:05AM AA.00006 Mean Flow Measurements with Hot Wires in High Reynolds Number Boundary Layers , R. DUNCAN, N. HUTCHINS, A. SEGALINI, J. MONTY, AND ICET TEAM — Mean flow measurements of high Reynolds number, Zero Pressure Gradient (ZPG) turbulent boundary layers are presented from the ICET data set using hot wire anemometry. The measurements were performed at momentum thickness Reynolds numbers in the range from 11,000 to 70,000, and compared to Pitot probe measurements at the same conditions. Various wire diameters, sensing lengths, probe designs and construction techniques are used, as well as different anemometer setups, in each of the facilities. Mean flow similarity between the three facilities is shown to be well within expected experimental uncertainty and ZPG layer manifestations, both when examining mean velocity profiles and integral parameters. The results reinforce the need for accurate near wall velocity and position measurements, as well as consistent analysis of physical and instrumentation biases. Various approaches are used to determine parameters such as the shape factor, the logarithmic overlap-region parameters, and the wake or outer flow parameters. Parameters extracted from the hot-wire profiles and those based on Pitot probe data are also compared and discussed in light of past experience with both instruments in different wall bounded flow experiments. Finally, consistency of the results is examined between the profile data and the skin friction behavior with Reynolds number, as measured by oil film interferometry.

9:18AM AA.00007 Turbulence Measurements with Hot Wires in High Reynolds Number Boundary Layers , J.H.M. FRANSSON, N. HUTCHINS, R. OERLUE, M. CHONG, AND ICET TEAM — During the last decade there has been a renewed interest in the scaling of turbulent boundary layers, especially with regard to the mean and fluctuation velocity distributions. Recently the ICET team carried out velocity measurements in three different wind tunnels (at KTH, Univ. Melbourne and IIT) for overlapping Reynolds numbers in the range $11,000 < Re_\theta < 70,000$. The use of different facilities enables measurements at similar Reynolds numbers, but with different free stream velocities (due to different development length for the boundary layer in the different wind tunnels). A number of different hot-wire probes and anemometers were used. In addition, accurate and independent skin friction measurements using oil film interferometry have been made to determine the friction velocity (u_τ), which is essential for accurate scaling of the data. The peak value of the near wall rms of the streamwise velocity was found to increase with Reynolds number, when scaled with u_τ. On the other hand, the skewness and flatness of the streamwise velocity are found to exhibit similarity in the near wall region if measured with sufficiently small (in viscous units) hot-wire probes, indicating a similarity of the probability density distributions independent of Reynolds number. The measurements also provide time series that are used to evaluate the scaling of spectra and other time-domain quantities.

9:31AM AA.00008 Study of turbulent boundary layer structures using Tomographic PIV[1] , QI GAO, ELLEN LONGMIRE, CECILIA ORTIZ-DUENAS, Aerospace Engineering and Mechanics, University of Minnesota — Tomographic-PIV was applied to investigate vortical structures in the logarithmic region of turbulent boundary layers. Measurements were performed in a water channel facility with $\delta \approx$110 mm for $Re_\tau \approx$2400 and 2900. Laser sheets with thickness up to 7mm were aligned parallel to the bounding surface. Four cameras with 2k x 2k pixels were placed in a rectangular array facing the measurement volume with tilt angle $\sim$30° to the wall normal direction. Magnification was $\sim$0.05 mm/pixel. The resulting measurement volumes were 0.8δ x 0.8δ in the streamwise and spanwise directions and 0.065δ or 120 viscous units in the wall-normal direction. Correlations were performed on 64^3 voxel volumes with 75% overlap yielding a vector spacing of 25^3 viscous units. The data were probed using swirl strength and direction as well as convection velocity to identify and characterize relatively large scale eddies and structures within the volumes. The results will be discussed and compared with results at similar wall-normal locations in lower Reynolds number DNS channel (Re_τ=590, 934 of Moser et al., 1999 and del Álamo et al., 2004) and wind tunnel (Re_τ=1160) flows.

[1]Supported by NSF (CTS-0324898).

9:44AM AA.00009 Tomographic Particle Image Velocimetry Measurements of a High Reynolds Number Turbulent Boundary Layer[1] , CALLUM ATKINSON, Laboratory for Turbulence Research in Aerospace and Combustion, Monash University, MICHEL STANISLAS, Laboratoire de Mecanique de Lille, Ecole Centrale de Lille, JULIO SORIA, Laboratory for Turbulence Research in Aerospace and Combustion, Monash University — Streamwise/wall-parallel volumes in the buffer region of a turbulent boundary layer at Re_θ =7800 and 11800 are measured using a 4 camera (2048 $\times$ 2048 px) tomographic particle image velocimetry (Tomo-PIV) system in the turbulent boundary layer wind tunnel at the Laboratoire de Mécanique de Lille (LML). Measurement volumes of 1200 $\times$ 180 $\times$ 1200 pixels are achieved, the large boundary layer provided by this tunnel ($\delta \sim$ 0.3 m) resulting in volumes of $470^+ \times 70^+ \times 470^+$ and $920^+ \times 140^+ \times 920^+$ wall units, respectively. The quality of the data acquired by this technique is assessed based on the mean velocity profile, velocity fluctuations, velocity power spectrum and the fluctuating divergence. Streaks and streamwise vortices are examined and an attempt is made to classify the flow using the invariants of the full velocity gradient tensor.

[1]Support of the Australian Research Council and the Eiffel Fellowship is gratefully acknowledged.

9:57AM AA.00010 Flow Structures and Effects of Spatial Resolution on Turbulence Statistics in Rough Wall Turbulent Channel Flow[1] , JOSEPH KATZ, JIARONG HONG, Johns Hopkins University, MICHAEL SCHULTZ, US Naval Academy — PIV data obtained in the roughness sublayer of a turbulent channel flow is used for examining effects of spatial resolution on the magnitude, distribution, and trends of Reynolds stresses. Starting with a vector spacing of 63um (9-12 wall units), for roughness consisting of 0.45mm high pyramids at Re_τ=3400-5418, spatial filtering of data causes major reduction in the magnitude of Reynolds stresses in the roughness sublayer. Although these reductions extend to well above the log layer, they increase with decreasing distance from the wall, especially for terms involving the wall-normal velocity fluctuation component, but also for the streamwise component. As expected, these effects increase with filter size, and are much higher for 2D filters in comparison to 1D ones. Consequently, trends of Reynolds stresses, and even mean flow profile vary significantly with filter properties. Spatial energy spectra and distributions of 2D swirling strength show the increasing role of small scale eddies on 2nd order statistics as the wall is approached, which is attenuated by filtering.

[1]Sponsored by ONR.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AB Turbulent Shear Layers 101B

8:00AM AB.00001 Evaluation of Similarity Theory for Weakly Swirling Jets using LDA Measurements , RICHARD SEMAAN, Univeristy of Wyoming, JONATHAN NAUGHTON, University of Wyoming — A similarity theory for weakly swirling jets developed by Ewing in 1999 is considered using recent experimental measurements. Momentum conserving jets with swirl numbers varying from 0 to 0.4 were measured using 3-D Laser Doppler Anemometry. This range ensured that none of the jets exhibited vortex breakdown. High quality measurements of the mean velocities and Reynolds stresses allow for a comprehensive evaluation of the theory. Despite the rapid decay of the swirl velocity relative to the axial velocity, the jet behavior is shown to be consistent with Ewing's theory even in the limited axial region where the swirl velocity is still significant. This is demonstrated through use of the experimental data to consider the constraints that are required for and the scaling quantities that result from the development of the similarity theory. The similarity theory governing weakly swirling jets is shown to hold for a range swirl numbers and thus provides another canonical flow for validating models. This theory is particularly useful for validating models that attempt to capture curvature effects.

8:13AM AB.00002 Coherent structures in uniformly sheared turbulence[1] , CHRISTINA VANDERWEL, STAVROS TAVOULARIS, University of Ottawa — Uniformly sheared flow with a turbulence Reynolds number Re_λ in the range from 100 to 140 has been generated in a water tunnel and its instantaneous structure has been examined using flow visualization, LDV and PIV. The flow was found to consist of regions with nearly uniform velocity and separated by relatively strong shear layers containing large vortices. The concentration of vortices and the probability distribution functions of their directions of rotation, strengths, sizes and shapes in three characteristic planes of the flow have been determined. These results, as well as high-speed scans of injected dye patterns, show that horseshoe-shaped vortices are prevalent in this flow.

[1]Financial support for this work was provided by NSERC.

8:26AM AB.00003 The Low Speed Intermittent Region of a Single Stream Shear Layer (SSSL)[1]

, JOHN FOSS, Michigan State Unversity, JASON PEABODY, Michigan State University — External intermittency: intermittent presence of vortical fluid, exists on the high and the low speed sides of a SSSL. From Corrsin and Kistler (1954) it is understood that the vortical fluid is bounded by a viscous superlayer (VSL) that is well-defined given zero vorticity in the entrained fluid. That condition is met for the SSSL flow of Morris and Foss (2003). The smoke trace from an incense stick provides a convenient technique to mark the 3-D locations of the VSL. Specifically, the vorticity at the VSL "shreds" the end of the trace; the shredding is captured by two cameras with calibrated x-y=f(z) and y-z=g(x) image planes. Complementary transverse vorticity measurements were acquired in the spatial domain indicated by the "shredding." Interpreted results will be presented. Corrsin, S. and Kistler, A.L. (1954) "The Free Stream Boundaries of Turbulent Flows," NACA TN 3133. Morris, S.C. and Foss, J.F. (2003) "Turbulent boundary layer to single-stream shear layer: the transition region", Jour. Fluid Mechanics, 494, pp. 187-221.

[1]Support from NSF, CTS0352710

8:39AM AB.00004 An Experimental Study of Circular and Rectangular Heated Jets in Cross-Flow

, B.E. JOHNSON, G. ELLIOTT, K.T. CHRISTENSEN, Univ. of Illinois — Heated plumes of fluid emanating from both flush and raised stacks occur in a wide variety of technologically-relevant applications and the dispersion of this heated, and sometimes contaminated, fluid is therefore of practical concern. To this end, a detailed series of experiments are under way to document the downstream development of heated jets of both circular and rectangular cross-section emanating into a cross-flow with the intent of constructing an experimental database for validation of on-going large-eddy simulations. Multiple cross-stream velocities as well as various jet exit velocities and temperatures are under study to understand the plume development both in the near- and far-field. The jet flow is driven by a high-pressure regenerative blower and heated with an open-coil heater while the cross-flow is provided by an open circuit wind tunnel. Mean temperature fields and streamwise velocity fields are measured using a rake of Pitot-static probes and thermocouples at select locations downstream of the jet. Of particular interest, the impact of jet exit temperature, velocity and cross-section on the downstream development of the heated plume is explored. Similarities and differences between flush and raised stack configurations under identical experimental conditions will also be highlighted.

8:52AM AB.00005 Open Loop Forced Experimental Investigation of Optical Beam Propagation through a Free Shear Layer

, CASEY FAGLEY, STEFAN SIEGEL, JURGEN SEIDEL, THOMAS MCLAUGHLIN, US Air Force Academy — The performance of airborne platforms emitting or receiving light beams is severely hampered by the flow field around the turret mounted on the air vehicle. From a fluid dynamics point of view, the flow separating from the turret develops large, coherent structures. From an optical point of view, these structures due to their associated density variations, cause large optical distortions since the index of refraction is a function of density. The goal of this research is to reduce optical distortions by mitigating these structures using feedback flow control. A blowing and suction slot along the top of the backwards facing step allows for actuation of the flow field. A study varying open loop forcing frequency and amplitude of the actuation signal shows the influence on the natural shedding frequency responsible for the large coherent structures. Initial findings support that forcing with more than twice the natural frequency does not produce structures that increase the optical path difference (OPD), measured by a Malley probe, while hot film measurements show that structures at the forcing frequency are present. The differences between these OPD and hot film measurements will be demonstrated and supported with simulation results.

9:05AM AB.00006 Spectral analysis of "homogenized" PIV data of axisymmetric jet turbulence

, MAJA WÄNSTRÖM, Chalmers University of Technology, WILLIAM K. GEORGE, CNRS, LML, Lille, France, KNUD ERIK MEYER, Technical University of Denmark — This paper concerns the estimation of one-dimensional spatial spectra from a set of 10,580 two-component velocity fields in a plane cut along the centerline axis of a turbulent air jet. The jet nozzle diameter $d = 1cm$ and the field-of-view extends from $30d$ to $100d$ and covers at least four jet half-widths. The Reynolds number based on nozzle diameter is $Re = 20,000$. Following [1], PIV data of the growing flow was mapped and re-sampled into streamwise homogeneous similarity coordinates, $\eta = r/\delta_{1/2}(x)$ and $\xi = \ln[(x - x_o)/D]$. Inevitable spatial variation in attainable dynamic range was overcome by the extensive number of statistically independent samples. Studies of the effects of window length and type, filtering, sampling noise and data re-sampling on the spectral energy distribution are performed and discussed. Finally, the spectral estimates are compared to the temporal estimates of [2], which are obtained by hotwire using Taylor's Hypothesis.

[1] Ewing, D. et al. J. Fluid Mech., 577, 309-330, 2007.
[2] Frohnapfel, B. Multi-Point Similarity of the Axisymmetric Turbulent Far Jet and Its Implication for the POD. Master Thesis, U. Erlangen 2003.

9:18AM AB.00007 ABSTRACT WITHDRAWN –

9:31AM AB.00008 Measurements in a High Reynolds Number Wake[1]

, JUAN JIMENEZ, MARCUS HULTMARK, ALEXANDER SMITS, Princeton University — Experiments were conducted in the Princeton/ONR HRTF windtunnel with air pressurized up to 220atm. The wake of a DARPA SUBOFF submarine model was measured at 5 different downstream locations for Reynolds numbers from 1×10^6 to 70×10^6. For all Reynolds numbers studied, the mean velocity distribution is self-similar from 3 diameters, D, downstream for the side where the support is not located. In contrast, self-similarity in the Reynolds stresses is not reached at the furthest downstream location ($x/D = 15$). The non-dimensional fluctuations are Reynolds number dependent for all measured Reynolds numbers. The energy spectra reveal two peaks in the near-wake. The lower wavenumber peak corresponds to a Strouhal number based on diameter and freestream velocity of about 0.2, suggesting that it is associated with an azimuthal or helical shedding mode in the wake. The peak decays with downstream distance, suggesting that this mode might play a parital role in the approach to self-similarity of the turbulent stresses

[1]Supported under ONR Grant N00014-07-1-0111, Program Manager Ron Joslin.

9:44AM AB.00009 Flow Field of Three-Dimensional Turbulent Wall Jets

, MARK TACHIE, MARTIN AGELIN-CHAAB, University of Manitoba — A wall jet is formed when a jet of fluid is directed tangentially along a wall. Wall jets can be two- or three-dimensional. Three-dimensional wall jets (3DWJs) are complex flows whose structures are still not well understood despite the extensive studies on this subject. For example, the mechanism responsible for their more rapid lateral spread rate than in the wall-normal spread rate is not well understood. Velocity measurements of 3DWJs were conducted using particle image velocimetry. The 3DWJs were formed by jets exiting a $d = 7$ mm inside diameter circular pipe ($143d$ in length) placed to flush the test section floor. The Reynolds numbers based on the jet exit velocities and jet exit diameters were 5000, 10000 and 20000. The detailed flow fields of the 3DWJs were examined in terms of mean velocities and one-point turbulence statistics. In view of the wide range of length and temporal scales that are present in turbulent flows, multi-point turbulence statistics such as two-point velocity correlations and proper orthogonal decomposition are used to document the salient features of the 3DWJs.

9:57AM AB.00010 Structure of three-dimensional turbulent offset jets with small offset distances , MARTIN AGELIN-CHAAB, MARK TACHIE, University of Manitoba — An offset jet is a jet that discharges into a medium above a wall which is offset by a certain distance. The "Coanda effect" forces the offset jet to deflect towards the wall and eventually attaches itself to the wall. The only detailed study of three-dimensional offset jets (3DOJs) did not report the flow field in the region from the jet exit to the point where the jet attaches itself to the wall. In this region flow reversal is expected. Velocity measurements of 3DOJs were conducted using particle image velocimetry. The 3DOJs have different jet exit offset distances (h) normalized by the jet exit diameter (d) of h/d = 0.5 to 4. The Reynolds numbers based on the jet exit velocities and jet exit diameters were 5000, 10000 and 20000. The detailed flow fields of the 3DOJs were examined in terms of mean velocities, and one-point turbulence statistics. In view of the wide range of length and temporal scales that are present in turbulent flows, multi-point turbulence statistics such as two-point velocity correlations and proper orthogonal decomposition are used to document the salient features of 3DOJs.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AC Turbulence Simulations I 101C

8:00AM AC.00001 Large-eddy simulation of the flow over two-dimensional dunes , M. OMIDYEGANEH, U. PIOMELLI, A.M. DA SILVA, Queen's University, Kingston, Canada — When a fluid flows over a mobile sand bed the sediment transport generated by the interaction of the flow field with the bed often results in the periodic deformation of the bed in the form of dunes. Dunes generally reach an equilibrium shape, and slowly propagate in the direction of the flow, as sand is lifted in the high-shear regions, and redeposited in the separated-flow area. Our aim is to attempt to connect the flow-field characteristics with the bed deformation. As a first step, we perform large eddy simulation of the flow over a typical dune geometry at laboratory scale (the Reynolds number based on the average channel height and mean velocity is 18,900). We consider three dunes, with different heights (relative to the channel depth) but equal wavelengths, using approximately ten million grid points. The mean flow shows a recirculation region downstream of the dune crest, whose extent increases with dune height. After reattachment the shear stress becomes high, confirming that sediment is lifted up in this region. The Reynolds stresses are higher in the shear layer, where the high spanwise vorticity gives rise to coherent vortices. The budgets of turbulent kinetic energy show that, in addition to production and dissipation, the diffusion terms play an important role. In the reattachment region, diffusion and dissipation are more significant. The mean flow advection is important at the beginning of the shear layer.

8:13AM AC.00002 Large Eddy Simulation study of the fully developed wind-turbine array boundary layer[1] , CHARLES MENEVEAU, Johns Hopkins University, MARC CALAF[2], Ecole Polytechnique Federale de Lausanne, Switzerland, JOHAN MEYERS, Katholieke Universiteit Leuven, Belgium — When wind turbines are deployed in large arrays, their efficiency decreases due to complex interactions among themselves and with the atmospheric boundary layer (ABL). For wind farms whose length far exceeds the height of the ABL, a fully developed flow regime can be established. Such a fully developed wind-turbine array boundary layer may be studied computationally using periodic boundary conditions in the horizontal direction. A suite of Large Eddy Simulations in which wind turbines are modeled using the classic "drag disk" concept are performed, in order to quantify the vertical transport of momentum and kinetic energy across the boundary layer. LES for various wind turbine arrangements, loading factors, and surface roughness are performed. Horizontally averaged statistics are documented. Results are compared with models for effective roughness length scales experienced by the ABL. This scale is often used to parameterize wind turbine arrays in models of atmospheric flow at regional or global scales. Based on the observed trends, a modified model is proposed showing improvements in the predicted effective roughness heights.

[1]Supported in part by NSF (Project CBET 0730922) .
[2]Visiting PhD student, Johns Hopkins University

8:26AM AC.00003 Coherent structures in a turbulent flow over an urban canopy made of cubical obstacles , STEFANO LEONARDI, University of Puerto Rico at Mayaguez, IAN CASTRO, University of Southampton UK — The study of turbulent heat or mass transport is of special interest in engineering, especially for heat exchangers. For instance, roughness elements (turbulators) are usually placed on the walls of the internal channels of a turbine blade to enhance the heat transfer. In the present paper, DNSs are carried out for passive heat transport in a turbulent channel flow with Λ shape square ribs for w/k = 1, 3, 7, 15 (w being the pitch, k the height of the ribs turbulators. The angle of inclination of the lambda shape turbulators is 45 degrees. Numerical results show that Λ shape square ribs are more efficient than square ribs in maximizing the heat transfer. The configuration with w/k=3 presents the largest heat flux. The increase in the heat transfer is due to a secondary motion which is generated by the Λ shape turbulators. Two counter rotating vortices above the square ribs transport the heat out of the wall into the center of the channel. The distribution of the heat flux coefficient is not uniform in the channel and leads to temperature gradients at the wall. The total drag of the Λ shape turbulators is larger than that over a smooth wall due to an increase of form drag.

8:39AM AC.00004 Direct numerical simulation of turbulent flow over a surface mounted obstacle[1] , NIKOLAOS MALAMATARIS, George Mason University — The direct numerical simulation of turbulent flow over a surface mounted obstacle is studied as a numerical experiment that takes place in a wind tunnel. For this reason, the incompressible, three dimensional, transient Navier-Stokes equations for Newtonian fluids are solved directly using Galerkin finite elements. The Reynolds number defined with respect to the height of the obstacle is in the range of 10^5. The results include instantaneous streamline patterns that show the vortex shedding phenomenon and the flapping of the recirculation bubble downstream the obstacle. Energy spectra are studied along with Eulerian autocorrelation coefficients, longitudinal and lateral coefficients that yield the chaotic behavior of turbulence. The computer code developed for this work is a parallel program written in Fortran 90 that uses the MPI-paradigm and runs in distributed memory systems. Movies are shown where both streaklines and instantaneous streamlines are depicted that clearly demonstrate the transient characteristics of this prototype separated flow.

[1]This research was supported in part by the NSF through TeraGrid resources provided by NICS.

8:52AM AC.00005 Large-eddy simulation of flow over a multi-element airfoil[1] , DONGHYUN YOU, Department of Mechanical Engineering, Carnegie Mellon University — An accurate prediction of turbulent flow over a multiple element high-lift airfoil configuration remains a challenge to computational fluid dynamics. Maximum lift, drag, and pitching moment are difficult to accurately predict especially in the presence of flow separation on one or more of the airfoil elements. In this study, we investigate turbulent flow over a MD30P30N high-lift configuration using large-eddy simulation. The MD30P30N configuration consists of three elements: a slat, a main airfoil, and a flap. Four different attack angles, 16°, 19°, 21°, and 24°, are considered while deflection angles of the slat and flap are fixed to 30°. The Reynolds number is 9×10^6 based on the mounted-wing chord-length and freestream velocity. Simulation results obtained on a 54 million-element mesh agree well with experimental data in terms of pressure distribution, velocity profiles, and transition location. A grid sensitivity study is performed to identify the resolution effects on the prediction of flow transition, wakes, and turbulent boundary layers. Accurate prediction of laminar-to-turbulence transition on the slat surface and downstream evolution of the slat wake is found to be crucial for the global accuracy of the simulation.

[1]Supported by the Boeing Company

9:05AM AC.00006 Numerical Simulation of Turbulence-Induced Bedform Initiation , YI-JU CHOU, OLIVER FRINGER, Environmental Fluid Mechanics Laboratory, Stanford University — Bedform initiation induced by near-wall turbulence structures on a sand bed is studied using large-eddy simulation. Due to the dilute sediment concentration during the simulation, transport of sediment is modeled with the Eulerian method. A second-order accurate arbitrary Lagrangian-Eulerian scheme is implemented that allows flow simulation over evolving bedforms. With a bed elevation model based on conservation of sediment mass to calculate changes in bed elevation, the present numerical model enables detailed observation of bedform instability caused by near-wall turbulence. It is found that the streak structure on the bed surface appears as the initial bed perturbation due to sediment erosion by turbulent sweeps, which in turn induce small pile-up at the downstream end of the inrush zone where the sweep diminishes. The continuous growth of the small sediment pile-up leads to the formation of bed defects, which alter the flow condition and the spatial distribution of near-bed sediment erosion. As a consequence, merging of multiple bed defects leads to the formation of ripple marks, which are a common bedform pattern in the subaqueous environment. The simulation results reveal interactions between turbulent structures and the sand bed and demonstrate the importance of the resolved turbulence in the simulation of bedform initiation.

9:18AM AC.00007 Direct numerical simulation of surface ablation by turbulent convection[1] , RYAN CROCKER, YVES DUBIEF, University of Vermont, CHRISTOPHER WHITE, University of New Hampshire — Rapid erosion by a turbulent flow creates complex flow/surface phenomena arising from the evolving surface topography and its interaction with a turbulent flow that transports the erosive agent onto the surface. The non-equilibrium nature of the problem poses major challenges to current turbulent models and boundary conditions used in direct numerical simulation (DNS) algorithms. A generalized algorithm for turbulent erosion processes based on level-set and immersed boundary methods has been developed in a DNS flow solver to investigate the action of various erosive agents (heat, particles, chemical species) on erodible surfaces. This algorithm is applied to the ablation of a slab of ice by natural and forced convection of water. The study focuses on the characterization of the surface topography in relation to the evolution of coherent structures in the flow, as ablation proceeds.

[1]R.C and Y.D. gratefully acknowledge the support of NASA Epscor (NNX07AT56A) and the computational resources provided by the Vermont Advanced Computing Center (supported by NASA-NNX-08AO96G).

9:31AM AC.00008 ABSTRACT WITHDRAWN —

9:44AM AC.00009 ABSTRACT WITHDRAWN —

9:57AM AC.00010 Comparing DNS and Experiments of Subcritical Flow Past an Isolated Surface Roughness Element[1] , CHARLES DOOLITTLE, DAVID GOLDSTEIN, University of Texas at Austin — Results are presented from computational and experimental studies of subcritical roughness within a Blasius boundary layer. This work stems from discrepancies presented by Stephani and Goldstein (AIAA Paper 2009-585) where DNS results did not agree with hot-wire measurements. The near wake regions of cylindrical surface roughness elements corresponding to roughness-based Reynolds numbers Re_k of about 202 are of specific concern. Laser-Doppler anemometry and flow visualization in water, as well as the same spectral DNS code used by Stephani and Goldstein are used to obtain both quantitative and qualitative comparisons with previous results. Conclusions regarding previous studies will be presented alongside discussion of current work including grid resolution studies and an examination of vorticity dynamics.

[1]Funded by AFOSR grant # FA-9550-08-1-0453.

Sunday, November 22, 2009 8:00AM - 10:10AM —

Session AD Flow Control I 101D

8:00AM AD.00001 Shock Wave/Boundary Layer Interaction Control in a Supersonic Inlet , NATHAN WEBB, EDGAR CARABALLO, JESSE LITTLE, The Ohio State University, JIN-HWA KIM, MO SAMIMY — A shock wave/boundary layer interaction (SWBLI) occurs in a supersonic mixed compression inlet. The SWBLI could cause boundary layer separation resulting in adverse consequences such as reduced pressure recovery and non-uniform fan loading. Boundary layer bleed is currently used to prevent separation, which incurs a significant performance penalty. We have recently used Localized Arc Filament Plasma Actuators (LAFPAs) with high amplitude and wide bandwidth to control the SWBLI in a Mach 1.9 flow. The preliminary results are promising and show excellent potential for this technique. These actuators may affect the SWBLI by two mechanisms: manipulation of flow instabilities and/or streamwise vorticity generation. Particle image velocimetry measurements have confirmed that instability manipulation is the key to the LAFPAs' ability to significantly energize the boundary layer in the interaction region. The streamwise vorticity effects are currently being investigated. The LAFPAs have been tested at varying frequency, duty cycle, and mode to determine the parameters with the maximum effectiveness. Supported by AFRL and AFOSR.

8:13AM AD.00002 ABSTRACT WITHDRAWN —

8:26AM AD.00003 Flow Mechanisms Leading to Separation Control over Embedded Hexagonal Cavities , AMY LANG, BLAKE MELNICK, University of Alabama — Digital Particle Image Velocimetry was used to measure the flow over an array of hexagonal cavities, with a focus on discerning the effect of cavity orientation on the adjacent boundary layer flow. Time-averaged velocity profiles above the cavities were measured under transitioning and turbulent boundary layer conditions. Two flow mechanisms leading to separation control, produced by the presence of the embedded cavities, were considered and will be discussed: (1) the presence of a partial slip velocity, produced by the embedded vortices forming within the cavities, on the adjacent boundary layer flow; and (2) turbulence augmentation close to the surface leading to a greater momentum exchange with the higher momentum, outer boundary layer region. The Reynolds stresses over the hexagonal cavities were thus compared to those over the flat plate under turbulent conditions to attempt to discern the effect of cavity orientation on turbulence augmentation. Results will discuss how these flow mechanisms lead to higher momentum in the boundary layer close to the wall as compared to a flat plate.

8:39AM AD.00004 Flow mechanisms induced by 2D transverse cavities leading to separation control[1]

, DREW SMITH, AMY LANG, University of Alabama, LEAH MENDELSON, Franklin W. Olin College of Engineering — Cavities embedded in the surface of an object moving through a fluid can help delay flow separation by imposing partial slip velocities and augmenting the turbulence of the boundary layer. Furthermore, recent experiments have shown that embedded cavity surface geometries may have their greatest effect on separation control at the point where the flow initially encounters them. This study investigated whether this effect is observed on a model with two-dimensional, transverse grooves embedded in the surface. The embedded cavities were mounted as a full-span section within a flat plate model in a low-speed water tunnel. Digital Particle Image Velocimetry was used to obtain flow data which were compared to that obtained over a flat surface. An analysis of the boundary layer profiles and Reynolds stresses at multiple locations on the model was conducted. Special attention was paid to the changes in these characteristics with streamwise distance.

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

8:52AM AD.00005 Long Lasting Modifications to Karman Vortex by a Single Pulse of DBD Plamsa[1]

, KWING-SO CHOI, TIM JUKES, University of Nottingham — We discovered a unique phenomenon whereby the vortex shedding and force fluctuations on a circular cylinder in cross flow are halted for a considerably long duration by applying a single, short-duration pulse of DBD plasma close to flow separation points. This period of flow modifications is equivalent to over 150 times that of the plasma excitation. We believe this is due to the induced vortex by a short-duration plasma pulse that interacts with the Karman vortex formation. As a result, the drag and lift fluctuations are reduced by 8% and 40%, respectively. This corresponds to the power-saving ratio of nearly 1200, or the energy efficiency of more than 50%. Experiments were conducted in a wind tunnel, where the free-stream velocity was 4.6 m/s and the turbulence intensity was 0.5%. The Reynolds number based on the diameter of circular cylinder was 15000. A single asymmetric DBD plasma actuator was placed on a circular cylinder at 75°, where a pulse of DBD plasma with a short duration (5% of the vortex shedding period) was applied at a high ac voltage (7.4 kV peak-to-peak, 33 kHz). Simultaneously with the force measurements using a two-component dynamic force balance, the global flow field in the near wake of a circular cylinder was studied using a time-resolved PIV system.

[1]Support by EPSRC Grant No. EP/D500850/1.

9:05AM AD.00006 Scalnig of transient lift response to actuation in a 3D separated flow[1]

, TIM COLONIUS, California Institute of Technology, DAVID WILLIAMS, Illinois Institute of Technology, GILEAD TADMOR, Northeastern University, WES KERSTENS, VIEN QUACH, SETH BUNTAIN, Illinois Institute of Technology — The transient lift response of a separated flow to short duration (pulsed) blowing is studied on a low Reynolds number, semicircular-planform, flat-plate wing. Actuators were distributed along the leading edge of the wing. The pulse duration, amplitude (supply pressure), and freestream speed were varied in the experiments. We identify two non-dimensional parameters governing the response, and use the data to find functional forms for the lift coefficient increment. We show that the lift coefficient increment is nearly independent of the pulse duration and increases (solely) with the square root of the supply-pressure coefficient up to a saturation. We also find that the shape of the lift response curve is similar to that produced in other experiments with different airfoils and actuators.

[1]Supported by AFOSR FA9550-05-1-0369.

9:18AM AD.00007 Optimized Control of Vortex Shedding from an Inclined Flat Plate[1]

, WON TAE JOE, TIM COLONIUS, DOUG MACMYNOWSKI, Caltech — Optimal control theory is combined with the numerical simulation of an incompressible viscous flow to control vortex shedding in order to maximize lift. A two-dimensional flat plate model is considered at a high angle of attack and a Reynolds number of 300. Actuation is provided by unsteady mass injection near the trailing edge and is modeled by a compact body force. The adjoint of the linearized perturbed equations is solved backwards in time to obtain the gradient of the lift to changes in actuation (the jet velocity), and this information is used to iteratively improve the controls. We investigate how features of the optimized waveform modify the vortex shedding and lead to higher lift, and compare the results with sinusoidal control. In order to obtain a practically implementable control scheme, the optimized waveform is also implemented in a simple closed-loop controller where the control signal is shifted or deformed periodically to adjust to the (instantaneous) frequency of the lift fluctuations. The feedback utilizes a narrowband filter and an Extended Kalman Filter to robustly estimate the phase of vortex shedding and achieve phase-locked, high lift flow states. Finally, the sensitivity of the flow to the phase shift and other features of the optimized waveform are presented.

[1]Work supported by AFOSR

9:31AM AD.00008 Separation Control in a 3D Diffuser using Plasma Actuators

, SVEN GRUNDMANN, JOHN K. EATON, Stanford University — Control experiments were conducted for the fully-turbulent flow in a 3D diffuser with an expansion ratio of 4.8. The uncontrolled flow for the same diffuser has a stable, three-dimensional separation zone which begins as a slender bubble in one corner before spreading across the entire width of the diffuser, giving the opportunity to develop and test active separation control devices. Dielectric-barrier discharge actuators were used to actively control the flow separation with the goal of improving the pressure recovery. The most effective control was achieved using spanwise acting plasma actuators in the inlet section of the diffuser which create streamwise vortices. The pressure recovery can be clearly improved or degraded depending on whether the actuators are operated pulsed or continuously. Parameter studies showed the dependence of the pressure recovery along the diffuser wall on the actuator operating parameters, including the modulation frequency and duty cycle. Velocity profile measurements in the inlet and outlet planes of the diffuser show the creation of the streamwise vortices and their influence on the uniformity of the velocity in the end of the diffuser. Frequency spectra taken in the exit plane using a hotwire probe show the influence of the operating parameters on the diffuser flow. A closed-loop control circuit for the automated adaption of the operating parameters is being tested.

9:44AM AD.00009 Separation Control on a Cascade of Airfoils using Pulsed Vortex Generator Jets[1]

, R.J. VOLINO, U. S. Naval Academy, M.B. IBRAHIM, O. KARTUZOVA, Cleveland State University — Flow through a row of airfoils will separate if the loading (lift) on each airfoil is too high. This can happen if the airfoil turning angle or the spacing between airfoils is too high. If the boundary layers separate, the actual flow turning and lift drop, and aerodynamic losses increase. In applications, such as the flow through turbines, high lift airfoils are desirable, as the same power generation can be achieved using fewer airfoils, thereby saving weight and cost. Advances in understanding of separation and transition have led to high lift airfoils without separation problems, but further increases in loading will likely require flow control. In the present study, flow through a linear cascade of very high lift low pressure turbine airfoils is controlled using pulsed vortex generator jets. Without flow control there is a large unclosed separation bubble at low Reynolds numbers. Separation causes a 20% drop in lift and increases losses by up to a factor of seven. Transition of the separated shear layer does not guarantee reattachment. Vortex generator jets with very low mass flow successfully control the separation if the jet velocity and pulsing frequency are sufficiently high. Experimental pressure distributions and phase averaged velocity and turbulence results will be presented.

[1]Supported by NASA

9:57AM AD.00010 Flow Separation Control over a High-lift Airfoil using Multiple DBD Plasma Actuators[1] , JESSE LITTLE, MUNETAKE NISHIHARA, IGOR ADAMOVICH, MO SAMIMY, The Ohio State University — This work continues an ongoing experimental study on the efficacy of plasma actuators for controlling flow separation on a high-lift airfoil. Previous results showed that a single dielectric barrier discharge (DBD) plasma actuator at the shoulder of a simple trailing edge flap can be effective for enhancing lift by increasing momentum transport between the freestream and separated region through amplification of large-scale structures at low frequencies. This work examines the ability of multiple actuators to further generate and amplify flow structures using pulsed actuation with variable phase. Multiple actuators are also operated at high frequency in an effort to reattach the separating boundary layer with quasi-steady plasma induced flow. Results show that low frequency pulsed forcing requires less power input and generates greater lift increases than high frequency actuation, but has a penalty of increased fluctuating pressure loads on the flap. These studies constitute necessary steps in the development and implementation of plasma actuators for control of flow separation at velocities associated with take-off and landing applications in transport aircraft.

[1]Supported by AFRL/DAGSI.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AE Biofluids I: General I - Biological Systems 101E

8:00AM AE.00001 Fluid Friction and Fungal Spore Ejection , JOERG FRITZ, AGNESE SEMINARA, School of Engineering and Applied Sciences, Harvard University, MARCUS ROPER, Department of Mathematics, University of California, Berkeley, ANNE PRINGLE, Department of Organismal Biology, Harvard University, MICHAEL BRENNER, School of Engineering and Applied Sciences, Harvard University — A wide range of fungal species in the phylum Ascomycota uses the forcible ejection of microscopic spores to disperse and to cover new territory, triggered by the breakdown of osmolytes in the sack containing the spores (the ascus). The spores experience very high aerodynamic drag due to their small size and need to attain high velocities to leave the boundary layer of still air around the fruiting body. Here we address the efficiency of conversion of osmotic pressure to the kinetic energy of the spore, and in particular its dependence on the design of the ascus and the hole (the so-called apical ring) from where the spores leave the ascus. We present a fluid mechanical model of the ejection process, which predicts that the hole the apical ring should have specific properties, in order to minimize frictional and pressure losses and maximize the ejection velocity. We compare these predictions to measurements of apical ring properties across the phylum.

8:13AM AE.00002 Dynamics of a nuclear invasion , MARCUS ROPER, Dept. of Mathematics, U.C. Berkeley, ANNA SIMONIN, N. LOUISE GLASS, Dept. of Plant and Microbial Biology, U.C. Berkeley — Filamentous fungi grow as a network of continuous interconnected tubes, containing nuclei that move freely through a shared cytoplasm. Wild fungi are frequently chimerical: two nuclei from the same physiological individual may be genetically different. Such internal diversity can arise either from spontaneous mutations during nuclear division, or by nuclear exchange when two individuals fuse, sharing their resources and organelles to become a single individual. This diversity is thought to be essential to adaptation in plant pathogens, allowing, for instance, an invading fungus to present many different genetic identities against its host's immune response. However, it is clear that the presence of multiple genetic lineages within the same physiological individual can also pose challenges - lineages that are present in growing hyphal tips will multiply preferentially. Nuclei must therefore be kept well mixed across a growing front. By applying models developed to describe mixing of fluids in microfluidic reactors to experimental observations of lineage mixing in a growing *Neurospora crassa* colony, we show how this mixing is achieved. In particular we analyze the individual contributions from interdigitation of hyphae and from nuclear transport.

8:26AM AE.00003 A model for wetting and evaporation of a post-blink precorneal tear film[1] , DANIEL ANDERSON, KATLYN WINTER, George Mason University, RICHARD BRAUN, University of Delaware — We examine a one-dimensional hydrodynamic model derived using lubrication theory for the evolution of a post-blink precorneal tear film that includes evaporation of the aqueous layer and a wetting corneal surface. The evaporation model includes the effects of conjoining pressure and predicts the existence of an equilibrium adsorbed fluid layer that serves as a model for a wetting corneal surface/mucin layer. The dewetting rates predicted by the model are in qualitative agreement with experimental measurements.

[1]NSF, DMS-0639300, DMS-0709095, DMS-0616483

8:39AM AE.00004 The ant raft , NATHAN MLOT, DAVID HU, SOLOMON EQUABAI, Georgia Institute of Technology — To survive floods, fire ants link their arms together to assemble a raft with their own bodies. Because ants are nearly as dense as water, this cooperative behavior requires that a portion of the ant colony must sacrifice itself by remaining underwater to support the colony's weight. Surprisingly, few ants drown during this process due to a striking metamorphosis of the raft: as we show using time-lapse photography, the raft morphs from a spherical to a pancake shape. This pancake configuration–a monolayer of floating ants supporting their dry counterparts–allows all ants to both breathe and remain united as a colony. Data is presented in the form of the dimensions and the rates of formation of the ant raft. We use the statics of small floating bodies to account for the equilibrium raft size as a function of the initial mass and density of the ants.

8:52AM AE.00005 Flight of the smallest insects , LAURA MILLER, ARVIND SANTHANAKRISHNAN, TYSON HEDRICK, University of North Carolina at Chapel Hill, ALICE ROBINSON, California Institute of Technology — A vast body of research has described the complexity of flight in insects ranging from the fruit fly, *Drosophila melanogaster*, to the hawk moth, *Manduca sexta*. Over this range of scales, flight aerodynamics as well as the relative lift and drag forces generated are surprisingly similar. The smallest flying insects (Re$\sim$10) have received far less attention, although previous work has shown that flight kinematics and aerodynamics can be significantly different. In this presentation, we have used a three-pronged approach that consists of measurements of flight kinematics in the tiny insect *Thysanoptera* (thrips), measurements of flow velocities using physical models, and direct numerical simulations to compute lift and drag forces. We find that drag forces can be an order of magnitude larger than lift forces, particularly during the clap and fling motion used by all tiny insects recorded to date.

9:05AM AE.00006 Legged locomotion on sand , CHEN LI, School of Physics, Georgia Institute of Technology, PAUL UMBANHOWAR, Department of Mechanical Engineering, Northwestern University, HALDUN KOMSUOGLU, DANIEL KODITSCHEK, Department of Electrical and Systems Engineering, University of Pennsylvania, DANIEL GOLDMAN, School of Physics, Georgia Institute of Technology — To understand how and why animals modulate foot kinematics to achieve effective locomotion on granular media, we study the speed of a six-legged robot with c-shaped legs, SandBot, moving on granular media for varying volume fraction, ϕ, limb frequency, f, and gait timing parameters[1]. Speed is determined by step length which in turn depends on limb penetration. At low f and high ϕ penetration is small, step length is large, and SandBot advances with a rotary walking gait in which c-legs rotate about their centers by slipping relative to stationary grains. In the opposite extreme, grains cannot support the robot; its underside always contacts the ground and it advances slowly via thrust generated as the c-legs translate through the grains. For varied gait parameters, high speeds are only observed in a small area of parameter space. A yield stress based model predicts the speed and reveals that performance is maximized when gait parameters minimize limb acceleration and interference, and limbs utilize the solidification properties of the media.

[1]Li et. al, PNAS, **106**, 3029, 2009

9:18AM AE.00007 Aquatic mapping using hydrodynamic pressure sensing , ROLAND BOUFFANAIS, DICK K.P. YUE, Massachusetts Institute of Technology — Pressure sensing is instrumental to most animals and organisms living in an aquatic environment: for instance fish at human scale through their lateral line and amoeba at microscale through mechanodetection at their surface. It also represents for underwater vehicles an alternative way of sensing the fluid environment when visual and acoustic sensing are limited. To assess the effectiveness of hydrodynamic sensing we propose a framework applicable to both high- and low-Reynolds number flows corresponding to typical fluid environment encountered by macro- and micro-swimmers respectively. In this framework both the forward and inverse problem corresponding to the object shape detection are presented. The forward mapping relies on a general solution of the pressure field expanded as an infinite series. The detection problem corresponds to the inverse problem which consists in determining some of the necessary coefficient of the expansion based on a noisy pressure signal over the limited length of the mechanosensing device.

9:31AM AE.00008 The hummingbird's tongue: a self-assembling syphon , JOHN BUSH, FRANCOIS PEAUDECERF, MIT, DAVID QUERE, ESPCI — We present the results of a combined experimental and theoretical investigation of the drinking technique of the hummingbird. Its long, thin tongue is dipped into nectar approximately 20 times per second. With each insertion, fluid rises along the length of the tongue through capillary action. While the tongue is open in cross-section, resembling a sliced straw, experiments demonstrate that surface tension serves to close it, with the tongue's zipping front corresponding to the rising meniscus. Supporting theoretical and analogue experimental models of this novel, natural example of capillary origami are developed and explored.

9:44AM AE.00009 Why Do Elephants Flap Their Ears? , MOISE KOFFI, LATIF JIJI, YIANNIS ANDREOPOULOS, City College of CUNY — It is estimated that a 4200 kg elephant generates as much as 5.12 kW of heat. How the elephant dissipates its metabolic heat and regulates its body temperature has been investigated during the past seven decades. Findings and conclusions differ sharply. The high rate of metabolic heat coupled with low surface area to volume ratio and the absence of sweat glands eliminate surface convection as the primary mechanism for heat removal. Noting that the elephant ears have high surface area to volume ratio and an extensive vascular network, ear flapping is thought to be the principal thermoregulatory mechanism. A computational and experimental program is carried out to examine flow and heat transfer characteristics. The ear is modeled as a uniformly heated oscillating rectangular plate. Our computational work involves a three-dimensional time dependent CFD code with heat transfer capabilities to obtain predictions of the flow field and surface temperature distributions. This information was used to design an experimental setup with a uniformly heated plate of size 0.2m x 0.3m oscillating at 1.6 cycles per second. Results show that surface temperature increases and reaches a steady periodic oscillation after a period of transient oscillation. The role of the vortices shed off the plate in heat transfer enhancement will be discussed.

9:57AM AE.00010 The effect of shear and flow separation on out of plane growth in biological films , DEREK RINDERKNECHT, MORY GHARIB, California Institute of Technology — Shear stress and flow separation are important physical cues initiating biofouling in many biological systems examples are the formation of plaques in the cardiovascular system and the accumulation of algae or other contaminants on the hulls of ships. To examine the effect of unsteady flow on the local shear profile and flow separation location and their relationship to the growth of thin biofilms, an experiment was constructed consisting of an open ended box with two opposing cylindrical half rounds located along the midline of the top and bottom faces. This chamber when mounted on a traverse is capable of creating steady, oscillatory and pulsatile flow profiles. A parametric study consisting of LIF dye experiments and PIV was conducted to examine the affect of unsteady flow amplitude and frequency on flow separation. Empirical velocity fields were analyzed using Lagrangian Coherent Structures to determine the impact of the unsteady flow profile on boundaries to transport within the flow. Results show the existence of three distinct flow regimes where the size and number of recirculations present depend on the frequency and amplitude of the oscillation. The flow was also seeded with algae and the apparent effects of flow separation and time periodic shear on out-of-plane biological growth will be discussed.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AF Microfluidics: Devices I 101F

8:00AM AF.00001 The hydrodynamic mobility of chiral colloidal aggregates , ERIC KEAVENY, MICHAEL SHELLEY, Courant Institute, New York University — A recent advance in colloidal technology [Zerrouki *et al.*, Nature **455**, 380 (2008)] uses magnetic aggregation to enable the formation of micron-scale particle clusters with helical symmetry from doublets composed of two micron-scale beads of different radii bonded together by a magnetic cement. Such self-assembled structures offer a means of controllable transport and separation in a low Reynolds number environment using externally applied magnetic or electric fields. We identify two necessary conditions that reveal further parameterized expressions that describe the positions of the beads in an aggregrate as a function of size ratio of the two beads composing the doublets. With the geometry of the structure known, we perform hydrodynamic calculations to ascertain entries of the mobility matrix for the aggregate and establish the relationship between the applied torque about the helical axis and translations parallel to this direction. For larger values of the particle radius ratio the coupling between rotations and translations changes sign as the number of doublets in the aggregate increases indicating that the clusters possess a more complex superhelical structure.

8:13AM AF.00002 Actuated cilial layers regulate deposition of microscopic solid particles , RAJAT GHOSH, Georgia Institute of Technology, GAVIN A. BUXTON, Robert Morris University, O. BERK USTA, ANNA C. BALAZS, University of Pittsburgh, ALEXANDER ALEXEEV, Georgia Institute of Technology — We use computational modeling to examine the three-dimensional interactions between oscillating, synthetic cilia and microscopic solid particles in a fluid-filled microchannel. The synthetic cilia are elastic filaments that are tethered to a substrate and are actuated by a sinusoidal force, which is applied to their free ends. The cilia are arranged in a square pattern and a neutrally buoyant particle is initially located between these filaments. Our computational studies reveal that depending on frequency of the beating cilia, the particle can be either driven downwards toward the substrate or driven upwards and expelled into the fluid above the cilial layer. This behavior mimics the performance of biological cilia used by certain marine animals to extract suspended food particles. The findings uncover a new route for controlling the deposition of microscopic particles in microfluidic devices.

8:26AM AF.00003 Size-based sorting of micro-particles using microbubble streaming , CHENG WANG, SHREYAS JALIKOP, SASCHA HILGENFELDT, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Oscillating microbubbles driven by ultrasound have shown great potential in microfluidic applications, such as transporting particles and promoting mixing [1-3]. The oscillations generate secondary steady streaming that can also trap particles. We use the streaming to develop a method of sorting particles of different sizes in an initially well-mixed solution. The solution is fed into a channel consisting of bubbles placed periodically along a side wall. When the bubbles are excited by an ultrasound piezo-electric transducer to produce steady streaming, the flow field is altered by the presence of the particles. This effect is dependent on particle size and results in size-based sorting of the particles. The effectiveness of the separation depends on the dimensions of the bubbles and particles as well as on the ultrasound frequency. Our experimental studies are aimed at a better understanding of the design and control of effective microfluidic separating devices. Ref: [1] P. Marmottant and S. Hilgenfeldt, Nature 423, 153 (2003). [2] P. Marmottant and S. Hilgenfeldt, Proc. Natl. Acad. Science USA, 101, 9523 (2004). [3] P. Marmottant, J.-P. Raven, H. Gardeniers, J. G. Bomer, and S. Hilgenfeldt, J. Fluid Mech., vol.568, 109 (2006).

8:39AM AF.00004 Whistling Bubbles: All-fluidic linear frequency sweep generators[1] , MANU PRAKASH, Harvard Society of Fellows, MATTHIAS KLAUSER, Massachusetts Institute of Technology — Complex biochemical systems are often composed of networks of interacting elements with spatial and temporal dynamics. Unlike ever so prevalent electronic debugging tools, automated tools to chemically probe these systems are in their infancy. Here we demonstrate one such novel tool - an all-fluidic frequency sweep generator capable of linear and exponential sweeps from a few Hz to KHz response. This is accomplished by coupling a flow-focusing geometry driven by negative pressure (- FF), a non-linear resistor and retraction dynamics of fluid threads in micro-channels. The device exhibits various modes including accordions and bursts with negative ramps to generate the desired sweep. The sweep parameters including f_{min}, f_{max} and T_{off} can be programmed by knobs including the hydrostatic pressure at inlet geometries. Our work highlights the importance of exploiting dynamics of drops and bubbles in fluidic networks to engineer desired function.

[1]We acknowledge Center for Bits and Atoms, MIT for use of experimental facilities and financial support from Defense Advanced Research Projects Agency (HROO11-09-C-0012) and Harvard Society of Fellows.

8:52AM AF.00005 High Speed motion generated by an oscillating microfiber , QI (RICK) HE, BIAN QIAN, Brown University, SHANE WOODY, BETHANY WOODY, Insitutec, KENNETH BREUER, Brown University — We present detail regarding the flow field generated by the high-speed motion of a long, thin elastic fiber immersed in a viscous fluid. The fiber, made of glass, or carbon, is approximately 1 mm in length, has a diameter of 7 microns, and is immersed in water, seeded with sub-micron tracer particles. The fiber is oscillated back and forth at 32 kHz with a peak-to-peak tip amplitude of approximately 10 microns. The resultant flow field is measured using micro-PIV, imaged at high speed using an intensified high speed camera, capable of taking data up to 12kHz. Symmetric vortices around the tip are generated by the steady streaming effect, with fluid velocities approaching 1 m/s, and shear rates close to $10^5\ s^{-1}$. The vortices have a strong three-dimensional structure due to the presence of the substrate below the moving tip, as well as the axial variation of the fiber amplitude. In addition to quantifying the fluid motion, the mixing of fluid and the dispersion, and accumulation of particles due to the fiber motion is measured and discussed.

9:05AM AF.00006 Multi - directional Electrophoretic Positioning of Charged Drops , B.S. HAMLIN, I.K. EMERSON, J.W. GAGE, W.D. RISTENPART, Univ. Calif. at Davis — We demonstrate a multi-directional electrophoretic technique to provide precise control over the position, trajectory and velocity of a charged object. Two or more pairs of electrodes, oriented along different directions, are periodically activated and deactivated to move the charged object along a desired trajectory. Two perpendicular sets of electrodes, for example, can induce the charged object to take a series of sudden 90° turns. We derive scaling expressions for the rates of acceleration and deceleration as the electric field direction is switched, and we corroborate the expressions experimentally for water droplets in oil. Surprisingly, we observe that the electrode geometry significantly alters the trajectory of charged drops, even in the absence of an applied electric field. The results have broad implications for any system that requires dynamic three-dimensional positioning, including lab-on-a-chip devices and electrophoretic displays.

9:18AM AF.00007 A theoretical model of electrophoretic biomolecule separation in periodic nanofilter arrays[1] , ZI RUI LI, NUS, NICOLAS HADJICONSTANTINOU, MIT, GUI RONG LIU, NUS, JONGYOON HAN, MIT, YU ZONG CHEN, JIAN-SHENG WANG, NUS — We present a theoretical model describing Ogston sieving—pore size comparable to or larger than the characteristic molecular dimension—of rigid isotropic and anisotropic biomolecules in nanofluidic filters, comprising of a periodic array of alternating deep and shallow regions. We obtain one-dimensional analytical results and two-dimensional numerical solutions of a drift-diffusion description, which captures the interplay between the driving electric force, entropic barrier and molecular diffusion. The analytical solution yields explicit results for the effective mobility and trapping time, and shows that the configurational entropy, which arises from the reduction of available configurations in the confined space of the nanochannel, dominates the resulting separation behavior. Our results are in line with experimental observations, and elucidate the effects of field strength, device geometry and entropic barrier height, providing a robust tool for the design and optimization of nanofilter/nanopore systems.

[1]This work was supported by the Singapore-MIT Alliance.

9:31AM AF.00008 Colon Cancer Cell Separation by Dielectrophoresis[1] , FANG YANG, XIAOMING YANG, H. JIANG, P. WOOD, W. HRUSHESKY, GUIREN WANG, NANO/MICRO FLUIDICS LAB, DEPT. ME UNIVERSITY OF SOUTH CAROLINA TEAM, CENTER FOR COLON CANCER RESEARCH, UNIVERSITY OF SOUTH, CAROLINA TEAM, GRACEFLOW TECHNOLOGY TEAM — Separation of cancer cells from the other biological cells can be useful for clinical cancer diagnosis and cancer treatment. In this presentation, conventional dielectrophoresis (c-DEP) is used in a microfluidic chip to manipulate and collect colorectal cancer HCT116 cell, which is doped with Human Embryonic Kidney 293 cells (HEK 293). It is noticed that, the HCT116 cell are deflected to a side channel from a main channel clearly by apply electric field at particular AC frequency band. This motion caused by negative DEP can be used to separate the cancer cell from others. In this manuscript, chip design, flow condition, the DEP spectrum of the cancer cell are reported respectively, and the separation and collection efficiency are investigated as well. The sorter is microfabricated using plastic laminate technology. -/abstract- This work has been financially supported by the NSF RII funding (EP

[1]This work has been financially supported by the NSF RII funding (EPS-0447660).

9:44AM AF.00009 Simulation of Taylor bubble flow in T-shaped micro-channel by MPS method , SHAOSHAN RONG, BIN CHEN, Xi'an Jiaotong University, KEY LAB. OF MULTI-PHASE FLOW IN POWER ENGINEERING TEAM — Moving Particle Semi-implicit (MPS) method uses particles and their interactions to simulate incompressible flow and it is a promising meshless method for multiphase flow simulation. In order to capture the interface in micro-scale channel, Taylor bubble flow in a T-shaped micro-channel is simulated in this paper. Firstly available surface tension and wettability model are improved and validated by simulating the droplet vibration and static shapes of a droplet attached on the solid surface, respectively. Afterwards, by discretizing the liquid and gas phases into moving particles with different density, bubble slug generation in T-shaped micro-channel is reproduced by MPS method with above models. The good agreement between numerical simulation with visualization experiment confirmed the capacity of MPS for the micro-scale two-phase flow. Finally, bubble generation mechanism is revealed by the velocity field of typical squeezing and shearing regime. The influences of viscosity, surface tension and contact angle on the bubble slug length are also discussed in detail.

9:57AM AF.00010 A Microscale Multi-Inlet Vortex Nanoprecipitation Reactor: Turbulence Measurement and Simulation[1] , JANINE CHUNGYIN CHENG, Dept. of Chemical and Biological Engr., Iowa State Univ., MICHAEL G. OLSEN, Dept. of Mechanical Engr., Iowa State Univ., RODNEY O. FOX, Dept. of Chemical and Biological Engr., Iowa State Univ. — Microreactors capable of generating turbulent flow are used in Flash NanoPrecipitation, a novel approach to produce functional nanoparticles. Microreactor design and optimization can be greatly enhanced by developing reliable computational models. Scalar mixing in a multi-inlet micro-vortex reactor has been previously studied and a RANS simulation with a scalar mixing model had been successfully validated for fully turbulent flow. To further validate the flow field, the reactor, operated in flow regimes from laminar to turbulent, was investigated using microscopic particle image velocimetry and computational fluid dynamics. In the experiments, instantaneous velocity fields were recorded on three different planes in the reactor and flow statistics such as the mean velocity and turbulent kinetic energy were computed for comparison with flow simulations. Laminar simulations were performed for the low-Reynolds-number cases, and large-eddy simulations (LES) were performed for higher-Reynolds-number cases. The simulation results were in good agreement with experiments for all cases, demonstrating the accuracy of the simulation models in the laminar-turbulent transition range.

[1]NSF CBET0403864 CBET0730250

Sunday, November 22, 2009 8:00AM - 9:57AM –
Session AG Free Surface Flows I 101G

8:00AM AG.00001 Resolving the unsteady deceleration and forces after water entry by low mass-ratio spheres , B.P. EPPS, MIT, T.T. TRUSCOTT, Naval Undersea Warfare Center, R.R. LAFOY, A.H. TECHET, MIT — The dynamics of water entry are significantly affected by surface coating and mass ratio, over a range of moderate impact velocities and sphere diameters. A hydrophobic sphere creates a sub-surface vapor cavity whereas a hydrophilic sphere does not, and the forces acting on the sphere after entry depend upon whether or not this cavity is formed. Using high speed-video, sub-pixel-accurate image processing techniques, and a smoothing spline method to find the derivatives of position data, we find the unsteady forces acting on spheres during the water entry event. Our data reveals that as mass ratio decreases from O(10 to 1), the sphere's deceleration becomes highly non-linear, since inertial forces cease to dominate over hydrodynamic forces. The unsteady deceleration is affected by vortex shedding in the no-cavity case, whereas in the cavity forming case, forces are affected by cavity growth and collapse, and vortex shedding is inhibited until after pinch-off. PIV sequences taken in both cavity-forming and non-cavity-forming impact cases highlight the vortex shedding onset and can be used to gain further insight into the sphere dynamics. Ultimately, this work emphasizes the need to accurately account for unsteady effects in modeling the post impact dynamics of spheres, especially as mass ratios approach unity.

8:13AM AG.00002 The water exit of buoyant spheres , TADD TRUSCOTT, Naval Undersea Warfare Center — Results of a combined experimental and theoretical investigation of a buoyant sphere as it passes through the free surface are presented. When a buoyant sphere ascends through a fluid column various behaviors are exhibited (e.g. vortex induced oscillations, etc.) and are a function of release depth and mass ratio. Using high speed dye visualization and particle image velocimetry to reveal the flow behavior, the conditions of release for which maximum height occurs and the governing parameters for optimum water exit are examined.

8:26AM AG.00003 Wave resistance , MARIE LE MERRER, FREDERIC CHEVY, ELIE RAPHAEL, CHRISTOPHE CLANET, DAVID QUERE, PMMH, ESPCI / LadHyx, Polytechnique — The more viscous, the more slippery! This is what happens when millimetric liquid nitrogen drops are thrown at the surface of water or viscous oil. Because these drops float on a cushion of vapor, the resistance to the motion mainly arises from the formation of waves, which mostly occurs on liquids of low viscosity. The wave resistance is very low, of the order of ten micronewtons. However, we could measure it from the slow deceleration of the drops as they move along the surface. We were able to show that this force increases very strongly (in a quasi-discontinuous fashion) when the velocity becomes higher than 23 cm/s, that is, when a stationary wake can exist. We also studied the effect of the bath viscosity, which damps the waves and hence reduces the drag.

8:39AM AG.00004 Laser-produced microjets , SIGURDUR THORODDSEN, King Abdullah University of Science and Technology, Saudi Arabia, K. TAKEHARA, T.G. ETOH, Kinki University, Osaka, Japan, C.-D. OHL, Nanyang Technological University, Singapore — We use ultra-high-speed imaging to characterize the formation of a micro-jet when a laser-produced shock hits a bubble sitting under a free surface. The bubble is formed inside a sessile drop, sitting on a glass slide and buoyancy drives it to its top. The jetting is forced by an Nd:YAG-laser pulse of about 30 mJ, focused by a microscope objective sitting under the glass plate. The jet is initiated when the shock hits the curved bottom of the bubble. It emerges out of a bottom crown and has a very regular shape. For water the jets are a few microns in size and can emerge at over 200 m/s. In intermediate viscosity liquids the jetting can be even faster and can emerge at over 500 m/s, depending on the depth of the laser focus. Jets can even be produced in pure glycerin where they emerge at about 100 m/s.

8:52AM AG.00005 Drag-out of bubbles by a plate withdrawn from a liquid bath[1] , JUSTIN KAO, ANDREA BLAKEMORE, ANETTE HOSOI, Massachusetts Institute of Technology — We report work on a new aspect of the classic Landau–Levich problem of liquid drag-out by a moving plate, namely, interaction of bubbles with the coating flow. Due to the cheerios effect, bubbles present in a liquid bath may gather at menisci such as those caused by a partially-submerged plate. Our experiments show that a critical withdrawal speed exists. Below this speed, bubbles are in stable equilibrium and remain stationary in the lab frame as the plate is withdrawn. Above the critical speed, no stable equilibrium exists and bubbles are drawn up onto the plate, with obvious consequences for the uniformity of the resulting coating. We examine the dependence of the critical speed on fluid properties.

[1]This work was supported by NSF DMS-0803083 and the MIT Undergraduate Research Opportunities Program.

9:05AM AG.00006 Laminar circular hydraulic jumps without separation , RATUL DASGUPTA, Jawaharlal Nehru Centre for Advanced Scientific Research, GAURAV TOMAR, University of California at Santa Barbara, RAMA GOVINDARAJAN, Jawaharlal Nehru Centre for Advanced Scientific Research — The traditional inviscid criterion for the occurrence of a planar, standing hydraulic jump is to have the Froude number decrease downstream and go through a value of 1 at some location. Here, upstream propagating, small-amplitude, long, non-dispersive gravity waves are trapped, and non-linear steepening is said to result in a near-discontinuous height profile, but it is not clear how. Such a condition on the Froude number is shown in the present axisymmetric Navier-Stokes computations to hold for a circular jump as well. The relevance of non-linear steepening to a circular jump is therefore a question we wish to answer. In circular jumps, moreover, a region of recirculation is usually observed underneath the jump, underlining the importance of viscosity in this process. This led Tani (J. Phys. Soc. Japan, 1949) to hypothesise that boundary-layer separation was the cause of the circular jump. This hypothesis has been debated extensively and the possibility of circular jumps without separation hinted at. In our simulations, we are able to obtain circular hydraulic jumps without any flow separation. This, and the necessity or otherwise of viscosity in jump formation will be discussed.

9:18AM AG.00007 Study of Air Entrainment by a Horizontal Plunging Liquid Jet , MARIO TRUJILLO, SURAJ DESHPANDE, University of Wisconsin, XIONGJUN WU, GEORGES CHAHINE, Dynaflow, Inc. — The process of air entrainment following the impact of an initially horizontal circular water jet on a pool of water has been studied computationally and experimentally. It has been found that the entrainment of air cavities in the near field region is periodic, not continuous as reported in earlier studies. The simulations are based on a Volume-of-Fluid methodology with interfacial compression using a modified version of the open source utilities, OpenFoam. Close agreement with experiments is reported on the creation of cavities in the near field, where air entrainment occurs. The period of entrainment is found to be proportional to g, and a simplified closed-form solution for this periodic event is presented. An overall physical picture of the mechanisms leading to bubble formation is given. The far field, which is characterized by the presence of small bubbles is only partially resolved computationally. Comparisons against velocity data are performed in this region leading to adequate qualitative agreement.

9:31AM AG.00008 Viscous rivulet flow over trenches[1] , THOMAS WARD, North Carolina State University, G.M. HOMSY, University of California Santa Barbara — The dynamic interfacial behavior of an advancing rivulet encountering a trench of square cross-section is studied experimentally at low capillary and Reynolds numbers. Trench depths vary from slightly smaller than to slightly larger than the capillary length. The fluids are a glycerol/water mixture and a silicone oil, representing a partially wetting and a nearly complete wetting fluid, respectively, and the rivulet interface is observed using low speed CCD imaging. A rich variety of phenomenon is observed in this range of depths suggesting that trench wetting is greatly affected by a combination of geometry and dynamic contact angle. The dynamics are characterized by measuring the local film height as a function of time and are compared with the theory of Gramlich et al. (Phys. Fluid, 2004). In spite of the fact that the theory is 2D while the experiments are 3D, remarkably good qualitative agreement is observed for large trench depths and partially wetting fluids.

[1]NC Space Grant

9:44AM AG.00009 Temporal statistics of a meandering rivulet[1] , PETER VOROBIEFF, The University of New Mexico, KEITH MERTENS, The University of North Carolina, Chapel Hill, VAKHTANG PUTKARADZE, Colorado State University — Meandering of a rivulet on an inclined, partially wetting surface can be triggered or suppressed by introducing small flow-rate fluctuations or, correspondingly, by reverting to a constant flow rate. Here we study a rivulet continuously meandering with flow-rate fluctuations present. Image sequences of the plane of the meander reveal a spatially and temporally resolved picture of the process, covering spatial scales from millimeters to meters and time scales from seconds to hours. For a given coordinate in the direction downstream from the origin of the rivulet, we construct time histories of the location of the rivulet centerline. Statistics of these time histories show an interesting correspondence with the spatial meandering statistics known from earlier work (such as the absence of a dominant wavelength), while also possibly manifesting previously unobserved trends at short time scales. It is also noteworthy that even a modest volume fraction of solid particles in the flow can radically alter the behavior of the rivulet, producing a stationary (pinned) meandering pattern.

[1]This research was partially supported by the National Science Foundation New Mexico EPSCoR award EPS-0447691.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AH Drops I: Impact 101H

8:00AM AH.00001 Thin Sheet Formation in Viscous Splash , MICHELLE DRISCOLL, SIDNEY NAGEL, James Franck Institute, University of Chicago — Ambient air is crucial for creating a splash on smooth dry surfaces for both viscous and inviscid liquids.[1] In a viscous splash, the drop initially spreads in the form of a thick lamella until t_{ejt} at which time it emits a thin fluid sheet. We have previously shown that t_{ejt} is set by the ambient pressure and the liquid viscosity, and shows only a weak dependence on drop impact velocity and surface tension.[2] We have measured the thickness of the ejected sheet using absorption measurements of a dyed liquid drop. The ejected sheet has a thickness $\sim 10\ \mu m$ that is approximately a tenth the thickness of the lamella preceding it. Using high-resolution, high-speed photography we have observed that as the ejected sheet expands, air bubbles are entrained into the trailing lamella. The bubble size increases as the lamella velocity decreases. Air entrainment ceases at a critical lamella velocity, $v_c \sim 1.2\ m/s$, which appears to be independent of drop impact velocity as well as the ambient pressure. At the critical velocity, the bubble radius is approximately 30 μm.

[1]L. Xu, *Phys. Rev. E* **75**, 056316 (2007); L. Xu *et al.*, *Phys. Rev. Lett.* **94**, 184505 (2005).
[2]M. Driscoll *et al.*, *DFD 2008* BAPS.2008.DFD.AG.5

8:13AM AH.00002 The dependence of Mesler entrainment on Weber number and drop axis ratio , J.R. SAYLOR, B.H. MILLS, F.Y. TESTIK, Clemson University — The impact of a water drop on a flat water surface can result in a variety of subsurface bubble formation events. Under certain conditions, the impact results in the formation of a large number of micron-scale bubbles, often referred to as Mesler entrainment. An experimental study is presented revealing that the existence of Mesler entrainment depends on both the drop Weber number and the drop axis ratio. Specifically, Mesler entrainment was observed for Weber numbers greater than 8 and less than 26. Within this range, the occurrence of Mesler entrainment was more frequent for axis ratios close to unity, that is for spherical drops. Drops of a prolate or oblate shape showed significantly less frequent Mesler entrainment. The working fluid for all experiments was water with a constant concentration of the soluble surfactant Triton X-100. This was done to avoid the influence of contaminating surfactants which tend to accrue when pure water is used as the working fluid.

8:26AM AH.00003 A computational study of high speed droplet impact , TOSHIYUKI SANADA, Shizuoka University, KEITA ANDO, TIM COLONIUS, California Institute of Technology — When a droplet impacts a solid surface at high speed, the contact periphery expands very quickly and liquid compressibility plays an important role in the initial dynamics and the formation of lateral jets. Impact results in high pressures that can damage the surface. In this study, we numerically investigated a high speed droplet impacts on a solid wall. The multicomponent Euler equations are computed by a FV-WENO scheme with an HLLC Riemann solver [Johnsen & Colonius, J. Comp. Phys. (2006)] that accurately captures shocks and interfaces. Stiffened equation of state is employed to model of gas, liquid and solid components. In order to compare the available theory and experiments, 1D, 2D and axisymmetric solutions are obtained. The generated pressures, shock speeds, and the lateral jetting mechanism are investigated. In addition, the effect of target compliance is evaluated.

8:39AM AH.00004 Events before droplet splashing on a solid surface , SHREYAS MANDRE, MADHAV MANI, MICHAEL BRENNER, Harvard University — A high velocity impact between a liquid droplet and a solid surface produces a splash. Classical observations traced the origin of this splash to a thin sheet of fluid ejected near the impact point, though the fluid mechanical mechanism leading to the sheet is not known. Mechanisms of sheet formation have heretofore relied on initial contact of the droplet and the surface. In this paper, we theoretically and numerically study the events within 1 μs of contact. The droplet initially tries to contact the substrate by either draining gas out of a thin layer or compressing it, with the local behavior described by a self similar solution of the governing equations. This similarity solution is not asymptotically consistent: forces that were initially negligible become relevant and dramatically change the behavior. Depending on the radius and impact velocity of the droplet, we show that the solution is overtaken by either the surface tension of the liquid–gas interface or viscous forces in the liquid. At low impact velocities surface tension stops the droplet from impacting the surface, whereas at higher velocities viscous forces become important before surface tension.

8:52AM AH.00005 Oscillating Effects on the Bubble Induced by A Free Falling Drop[1] , AN-BANG WANG[2], C.-C. KUAN, P.-H. TSAI, Institute of Applied Mechanics, National Taiwan University — The impact of a droplet on a liquid pool can result in different fantastic phenomena. Many investigations have been conducted since decades; however, none has been studied for the effects of oscillating drop on the big bubble induced by the impacting droplet since Worthington (1908). In the present study, big bubble induced by the droplet impact has been experimentally studied and systematically analyzed. Effects of impact velocity, drop size, oscillation parameters and depth of target liquid have been investigated and discussed. New characteristic regimes in the V (impact velocity)-d (diameter of droplet)-map have been discovered. Two geometry parameter oscillation parameters sharpness-ratio and offset-ratio of the free-falling droplet have been found to be the most important controlling parameters. Their results are revealed and compared in this study.

[1]Financial supports of NSC 97-2221-E-002-221-MY3 & 95-2221-E-002-341-MY3 are acknowledged.
[2]correspondance

9:05AM AH.00006 Drop impact on sand: from donut to pie , GILES DELON, GRASP-Optofluidics, STEPHANE DORBOLO, NICOLAS VANDEWALLE, GRASP, HERVE CAPS, GRASP-Optofluidics — We have studied the impact of water drops onto granular layers. Depending on the impact energy, various shapes are observed for the resulting craters. Experimental parameters that have been considered are : the size of the millimetric droplets; the height of the free fall, ranging from 1.5 cm to 100 cm; and, the depth of the granular layers, ranging from tenth of millimeters to a few centimeters. As the drop is impacting the sand layer, energy is dissipated and a splash of sand occurs. Meanwhile, surface tension, inertia and viscosity compete, leading to strong deformations of the drop which depend on the experimental conditions. Just after the drop enters into contact with the sand, imbibition takes place and increases the apparent viscosity of the fluid. Soon, the drop motion is stopped by this process. Images and fast-video recordings of the impact allowed us to draw scaling laws for the crater morphology and size.

9:18AM AH.00007 Suppressing Viscous Drop Splashing with Surface Roughness , ARIANA STRANDBURG-PESHKIN, Swarthmore College, MICHELLE DRISCOLL, SIDNEY NAGEL, James Franck Institute, University of Chicago — The splashing of a liquid drop on a smooth, dry surface depends on a host of factors: the speed, surface tension, viscosity and size of the drop, but also, surprisingly, the pressure and molecular weight of the surrounding gas.[1,2] In the case of a viscous drop splashing on a smooth surface, a thin sheet of fluid is first ejected from the rim of the expanding drop and then breaks up into droplets to form a splash.[3] When the surface is rough, different behavior, known as prompt splashing, may also be observed.[2,4] Here we explore the splashing of a viscous liquid as the surface roughness, R_a, is varied. We find that a small degree of roughness, $R_a < 1\ \mu m$, can completely suppress the thin-sheet ejection occurring on smooth surfaces. The degree of roughness necessary for this suppression decreases with increasing viscosity. In some cases, the roughness is great enough to suppress the thin sheet, but insufficient to produce a prompt splash, thus suppressing the splash entirely.

[1] L. Xu, W.W. Zhang, S.R. Nagel, PRL 94, 184505 (2005).
[2] L. Xu, PRE 75, 056316 (2007).
[3]M. Driscoll et al., BAPS DFD AG.00005 (2008).
[4] L. Xu, L. Barcos, and S.R. Nagel, PRE 76 066311 (2007).

9:31AM AH.00008 The effects of wind on the impact of a single drop on a water surface[1] , XINAN LIU, University of Maryland — The impact of single water drops on a water surface was studied experimentally in a wind tunnel. Water drops were generated from a needle oriented vertically from the top surface of the wind tunnel test section. 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, eventually hit a shallow pool of water on the bottom of the test section. The drop impacts were backlit with a halogen lamp and photographed with a high-speed movie camera 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 drops are formed and move in the leeward direction. This is followed by a stalk formation at the location of the water drop impact. It is found that the shape of the secondary-drop chain and the appearance of the stalk are markedly affected by wind speed. The effects of wind speed and initial drop size on a number of parameters, including the number, diameter and total mass of secondary drops were investigated. The dynamics of secondary drops in the presence of wind are discussed.

[1]This project is supported by NSF.

9:44AM AH.00009 Droplet impact on a porous substrate: a capillary tube model , HANG DING, THEO THEOFANOUS, CRSS, University of California, Santa Barbara — The dynamics of impacting (spreading, penetrating) a droplet on a porous substrate, modeled by an array of capillary tubes, is studied numerically using diffuse interface methods. The absorption rate depends on the diameter ratio of the capillary tube to the droplet, wettability, and liquid properties. The flow dynamics is resolved by solving the Navier-Stokes equations and interface capturing is governed by the Cahn-Hilliard equation. Contact-angle hysteresis is included (Ding&Spelt 2008) and the stress singularity at moving contact lines is relieved using a diffuse interface model (Seppecher 1996; Jaqcmin 2000). The model is validated by studying the evolution of a droplet initially resting on a porous substrate and by comparison to drop-impact experiments involving just one capillary tube (Kogan et al 2008). Comparisons with analytical solutions and results available in the literature (e.g. Hilpert & Ben-David 2009) are presented. Through parametric simulations over relevant ranges of Reynolds and Ohnesorge numbers and contact angles, impact regime maps are derived.

9:57AM AH.00010 Impact of Droplets on a Vertical Capillary Tube , ALINE DELBOS, ELISE LORENCEAU, OLIVIER PITOIS, MICHÈLE ADLER, LPMDI, Université Paris-Est — We experimentally study whether it is possible to force liquid impregnation of a porous media using the kinetics energy of an impacting drop. We study impregnation at the local scale and only consider a unique pore, a vertical glass capillary tube with thick walls. The forced impregnation is achieved with droplet of water impacting with an initial velocity on the tube. We focus both on forced impregnation of hydrophilic or hydrophobic capillary tubes. For small impact velocities, the classical results of impregnation are recovered. For large impact velocities, we observe new regimes due to the initial kinetics energy of the droplet. In particular, a liquid index disconnected from the upper part of the drop, which spreads on the horizontal flat surface, is observed both for the hydrophilic and hydrophobic tubes. To quantify the efficiency of the forced impregnation, we answer the following questions i) what is the volume of liquid eventually trapped within the porosity? ii) How deep this liquid is located in the pore?

Sunday, November 22, 2009 8:00AM - 9:57AM –
Session AJ Bubbles I: Cavitation 101I

8:00AM AJ.00001 Numerical simulation of shock/bubble-cloud interaction problems[1] , KEITA ANDO, California Institute of Technology, TIM COLONIUS, CHRISTOPHER BRENNEN, Califonia Institute of Technology — The interaction of a shock wave with a dilute bubble cloud is computed using a continuum two-phase model incorporating the effect of a distribution of nuclei sizes. The bubble dynamics are evaluated using a Rayleigh-Plesset-type equation including the effects of heat transfer, liquid viscosity and compressibility. A finite-volume WENO scheme coupled with an approximate HLLC Riemann solver is developed to solve the shock problems. Linear and shock wave propagation through a one-dimensional bubble screen is computed and the effect of phase cancellations among the different-sized bubbles is quantified. The size distribution in the screen is found to increase the cushioning of the shock loading. Computations of shock/bubble-cloud interaction in two dimensions are also presented.

[1]Supported by ONR Grant No. N00014-06-1-0730 and by NIH Grant No. PO1 DK043881.

8:13AM AJ.00002 Numerical Simulations of Bubble Dispersion over a Hydrofoil , SHUANG ZHU, ANDREW OOI, The University of Melbourne, HUGH BLACKBURN, Monash University, BRENDON ANDERSON, Defence Science and Technology Organisation — The production and entrainment of bubbles in ship wakes is not completely understood despite the fact that it has many practical applications. For example, bubbles trapped in the large vortical structures in the ship wake can form clusters that are able to persist for large distances leaving a long trail of bubbles, which increases the ship's signature; an important consideration in the defence environment. The fundamental mechanisms behind the complicated bubbly flow can be understood using data from numerical simulations. The objective of the study is to investigate the accuracy of current state-of-art numerical models for simulating bubbly flows. A spectral element-Fourier code will be used to carry out direct numerical simulations (DNS) with Lagrangian particle tracking to study the interaction of the upstream bubble distribution with a hydrofoil at different angles of attack and Reynolds numbers, and the effect on the resulting downstream bubble distribution.

8:26AM AJ.00003 Partial Cavity Flows at High Reynolds Numbers , SIMO MAKIHARJU, BRIAN ELBING, University of Michigan, ANDREW WIGGINS, DAVID DOWLING, MARC PERLIN, STEVEN CECCIO, University of Michigan — Partial cavity flows created for friction drag reduction were examined on a large-scale. Partial cavities were investigated at Reynolds numbers up to 120 million, and stable cavities with frictional drag reduction of more than 95% were attained at optimal conditions. The model used was a 3 m wide and 12 m long flat plate with a plenum on the bottom. To create the partial cavity, air was injected at the base of an 18 cm backwards-facing step 2.1 m from the leading edge. The geometry at the cavity closure was varied for different flow speeds to optimize the closure of the cavity. Cavity gas flux, thickness, frictional loads, and cavity pressures were measured over a range of flow speeds and air injection fluxes. High-speed video was used extensively to investigate the unsteady three dimensional cavity closure, the overall cavity shape and oscillations.

8:39AM AJ.00004 Ventilated Supercavities[1] , ELLISON KAWAKAMI, University of Minnesota, ROGER ARNDT, Uiversity of Minnesota — The topic of supercavitation is of considerable interest to drag reduction and/or speed augmentation in marine vehicles. Supercavitating vehicles need to be supplied with an artificial cavity through ventilation until they accelerate to conditions at which a natural supercavity can be sustained. A study has been carried out in the high-speed water tunnel at St. Anthony Falls Laboratory to investigate some aspects of the flow physics of such a supercavitating vehicle. During the present experimental work, the ventilated supercavity formed behind a sharp-edged disk was investigated utilizing several different configurations. Results regarding cavity shape, cavity closure and ventilation requirements versus cavitation number and Froude number are presented. Additionally, effects related to flow choking in a water tunnel test section are discussed. Results obtained are similar in character to previously reported results, but differ significantly in measured values. Cavity shape, particularly aft of the maximum cavity diameter, is found to be a strong function of the model support scheme chosen.

[1]Supported by ONR.

8:52AM AJ.00005 Effect of the cavity closure condition on the flow of liquid around a supercavitating wedge[1] , ANNA ZEMLYANOVA, YURI ANTIPOV, Louisiana State University — The problem for a one non-symmetric supercavitating wedge in a jet is considered. The single- and double-spiral-vortex models proposed by Tulin are used to describe the flow of the liquid at the rear part of the cavity. Both problems are solved in a closed form using the methods of complex analysis. The models are compared with respect to different parameters of the flow. It is obtained that the flow around the wedge, in the front part of the cavity and the lift and drag coefficients are not affected by the choice of the model. On the other hand, the flow at the tail part of the cavity and the length of the cavity depend strongly on the chosen model.

[1]This work was funded by NSF through grant DMS0707724.

9:05AM AJ.00006 Tip vortex cavitation suppression via mass injection , HARISH GANESH, University of Michigan, Ann Arbor, NATASHA CHANG, RYO YAKUSHIJI, STEVEN CECCIO, University of Michigan, Ann Arbor — Tip vortex cavitation (TVC) suppression by mass injection in the core of the vortex was studied with an elliptical plan-form hydrofoil NACA-66 modified in a re-circulating water tunnel of known nuclei distribution. The chord based Re was O(106) for all experiments. Water and Polyox WSR 301 solution for a range of concentrations (10 to 500pmm) and relative flow rates (Qjet / Qcore of 0.033 to 0.27) were injected. Also, different injection port size and angle of attack were studied. It was found that the TVC suppression effect was different for inception and desinence. The baseline (no injection) inception cavitation number was more than the average negative pressure coefficient, -Cp of the vortex, while mass addition reduced the inception cavitation number to approximately the –Cp value. TVC desinence for the baseline case was found to match the estimated –Cp value and polymer injection provided some cavitation suppression. Flow measurements were made to understand the underlying physics of TVC. The mechanisms and scalability that lead to TVC suppression by mass injection are discussed.

9:18AM AJ.00007 Dynamic of cavitation bubble in a flowing liquid with a pressure gradient[1] , MARC TINGUELY, MOHAMED FARHAT, Ecole Polyechnique Federale de Lausanne — In the present study, a high energy pulsed laser is used to generate a millimetric cavitation bubble within a water flow over a symmetric hydrofoil. The bubble is initiated at different locations in the vicinity of the hydrofoil leading edge. A high speed camera is used to observe the motion of the bubble as it travels along the hydrofoil suction side. Besides the standoff parameter, we have found that the pressure gradient plays a major role on bubble dynamic and subsequent phenomena. For a specific initial location of the bubble, the micro-jet is no more directed towards the hydrofoil surface, as commonly observed in still water. In this case, we have also observed a spectacular behaviour of the cavity rebound, which migrates towards the solid surface despite of the outward direction of the micro-jet. This result differs from the behaviour of a bubble near a solid surface in water at rest or water flowing uniformly since the micro-jet is normally directed toward the solid.

[1]Supported by Swiss National Science Foundation (Grant No 200020-116641).

9:31AM AJ.00008 Cavitation induced by high speed impact of a solid surface on a liquid jet[1] , MOHAMED FARHAT, MARC TINGUELY, MATHIEU ROUVINEZ, Ecole Polyechnique Federale de Lausanne — A solid surface may suffer from severe erosion if it impacts a liquid jet at high speed. The physics behind the erosion process remains unclear. In the present study, we have investigated the impact of a gun bullet on a laminar water jet with the help of a high speed camera. The bullet has a flat front and 11 mm diameter, which is half of jet diameter. The impact speed was varied between 200 and 500 ms^{-1}. Immediately after the impact, a systematic shock wave and high speed jetting were observed. As the compression waves reflect on the jet boundary, a spectacular number of vapour cavities are generated within the jet. Depending on the bullet velocity, these cavities may grow and collapse violently on the bullet surface with a risk of cavitation erosion. We strongly believe that this transient cavitation is the main cause of erosion observed in many industrial applications such as Pelton turbines.

[1]Supported by Swiss National Science Foundation (Grant No 200020-116641). With the collaboration of ARMASUISSE.

9:44AM AJ.00009 Cavitation Bubble in Shear Flow , SADEGH DABIRI, Massachusetts Institute of Technology, WILLIAM SIRIGNANO, University of California Irvine, DANIEL JOSEPH, University of Minnesota, University of California Irvine — In the orifice of liquid injectors at high pressure, cavitation occurs behind the sharp corners, where a strong pressure drop is present due to quick change in the flow direction. In addition, a high level of shear is present inside the boundary layer. Therefore, it is important to understand the influence of the shear on the cavitation. In this study, the deformation of a cavitation bubble in shear and extensional flows is numerically investigated. The Navier-Stokes equations are solved to observe the three-dimensional behavior of the bubble as it grows and collapses. During the collapse phase of the bubble, two re-entrant jets are observed on two sides of the bubble due to interaction of the bubble with the background flow. Re-entrant jets with enough strength could breakup the bubble into smaller bubbles. Post processing of the results is done to cast the disturbance by the bubble on the liquid velocity field in terms of spherical harmonics. It is found that a quadrupole moment is created in addition to the monopole source. As the bubble collapses regions of high vorticity are created near the bubble interface.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AK Multiphase Flows I 101J

8:00AM AK.00001 Dynamic Characteristics at the Interface of Underwater Round Gas Jets , CHRIS WEILAND, PAVLOS VLACHOS, Virginia Tech — The gas-liquid interface characteristics of round gas-jets submerged in water was studied across a wide range of Mach numbers (0.4-1.9). High speed shadowphotography was used to image the gas jet and the interface was tracked from the digital images for all points in space and time. The results show how the interface characteristics are governed by buoyancy to momentum driven flow as the Mach number increases. The jet penetration, defined as the maximum length a continuous gas jet occupies 99 percent of the time, increases with the injection Mach number. The penetration is related to the compressible jetting length, defined as the distance from the orifice where the momentum and buoyancy forces are balanced, and signifies a change in the jet behavior spatially from a momentum to buoyancy driven flow. The interface motion is computed as a function of the Mach number and the distance downstream from the orifice. These results indicate the most unsteady jetting process near the orifice occurs at Mach 1, presumably due to the formation of a shock cell structures.

8:13AM AK.00002 A numerical study of air layer drag reduction phenomenon on a flat plate[1] , DOKYUN KIM, PARVIZ MOIN, CTR, Stanford University — The objective of the present study is to predict and understand the air layer drag reduction (ALDR) phenomenon. Recent experiments (Elbing et al. JFM 2008) have shown large net drag reductions if air is injected beyond a critical rate at the wall. The stability analysis and numerical simulations are performed to investigate mechanisms of ALDR on a flat plate using the same geometry as in the experiment. The linear stability of air-liquid interface is investigated by solving the Orr-Sommerfeld equations, and numerical simulations of two-phase flow have been performed to describe the evolution of air-water interface. The stability analysis shows that the air flow rates, Reynolds number, Weber number, and Froude number are important parameters determining the stability of the air layer. In laminar boundary layer, it is observed from the numerical simulations that the Froude number is the key to the stability of the air layer. The presentation will include a new and very efficient numerical method for two-phase flow calculations used in this study.

[1]Supported by the Office of Naval Research.

8:26AM AK.00003 Lagrangian statistics for bubbles in a turbulent boundary layer[1] , MICHAEL MATTSON, KRISHNAN MAHESH, University of Minnesota — We are developing the simulation capability for 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. For this talk, direct numerical simulation is used to solve the Navier–Stokes equations for a spatially-evolving turbulent boundary layer ($Re_\theta = 1430$) and bubbles are injected into the near-wall region. The bubbly suspension is dilute and one-way coupled equations are used. The temporal evolution of the bubble dispersion and probability density functions of the bubble forces will be presented, with emphasis on the role Stokes number and injection location play in determining bubble behavior.

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

8:39AM AK.00004 A comprehensive subgrid air entrainment model for Reynolds-averaged simulations of free-surface bubbly flows , JINGSEN MA, ASSAD A. OBERAI, DONALD A. DREW, RICHARD T. LAHEY, JR, Rensselaer Polytechnic Institute, MARK C. HYMAN, Naval Surface Warfare Center–Panama City — The simulation of free surface bubbly flows using a two-fluid model remains challenging in part due to the lack of a comprehensive air entrainment model that can predict the location and rate of air entrainment for a wide range of flows. In this study we derive one such model and implement it into a computational multiphase fluid dynamics (CMFD) framework that solves the Reynolds-averaged two-fluid equations. The subgrid air entrainment model is derived from a simple argument that the wave action near the air/water interface causes it to ingest air bubbles and they are entrained into the liquid if their downward velocity exceeds that of the interface. This yields a simple expression for the rate of entrainment as a product of the downward gradient of the liquid velocity near the free surface and the turbulent kinetic energy. We have tested the performance of this model and CMFD in simulating the bubbly flow due to a plunging liquid jet, in a hydraulic jump and around a full-scale naval surface ship by comparing with experimental data. It's found that in each case the subgrid air entrainment model and the two-fluid modeling approach yields accurate results.

8:52AM AK.00005 On bubble clustering and energy spectra in pseudo-turbulence[1] , JULIAN MARTINEZ MERCADO, DANIEL CHEHATA GOMEZ, DENNIS VAN GILS, CHAO SUN, DETLEF LOHSE, Physics of Fluids Group, Faculty of Science and Technology, University of Twente — We performed 3D-Particle Tracking (3D-PTV) and Phase Sensitive Constant Temperature Anemometry in pseudo-turbulence to investigate bubble clustering and to obtain the mean bubble rise velocity, distributions of bubble velocities, and energy spectra at dilute gas concentrations. To characterize the clustering the pair correlation function $G(r, \theta)$ is calculated. The deformable bubbles with equivalent bubble diameter $d_b = 4 - 5$ mm are found to cluster within a radial distance of a few bubble radii with a preferred vertical orientation. This vertical alignment is present at both small and large scales. The large number of data-points and the non intrusiveness of PTV allowed to obtain well-converged Probability Density Functions (PDFs) of the bubble velocity. The PDFs have a non-Gaussian form for all velocity components and intermittency effects can be observed. The energy spectrum of the liquid fluctuations decays with a power law of -3.2, different from the $\approx -5/3$ found for homogeneous isotropic turbulence.

[1]This research was funded by the Stichting voor Fundamenteel Onderzoek der Materie (FOM).

9:05AM AK.00006 How does interfacial rheology govern soap bubble cluster dynamics? , SYLVIE COHEN-ADDAD, Universite Paris-Est, ANNE-LAURE BIANCE, Universite Lyon 1, REINHARD HOHLER, Universite Paris-Est — Aqueous foams are concentrated dispersions of gas bubbles in a soapy solution. These complex fluids exhibit solid-like or liquid-like mechanical behaviors, depending on the applied shear. When it is increased beyond a yield strain, neighbor switching bubble rearrangements called T1 events are triggered and plastic flow sets in. We study experimentally the dynamics of such strain induced T1s in 3D bubble clusters that we consider as model systems of 3D foams. To determine the hydrodynamics and physico-chemistry that set the duration of T1s, we use foaming solutions of a wide range of well characterized bulk and interfacial rheological properties. At low shear rates, the T1 duration is set by a balance between surface tension and surface viscous forces in qualitative agreement with previous studies of T1s in 2D foams [1] and we present a simple physical model that explains our 3D findings. Moreover, above a characteristic shear rate, rearrangement dynamics are driven by the applied strain. By combining all our results, we link the transition from intermittent to continous flow dynamics in foams to the rheology of the gas-liquid interfaces.

[1] M. Durand, H. A. Stone, *Phys. Rev. Lett.* **97**, 2226101 (2006).

9:18AM AK.00007 Measurements of the fluctuating liquid velocity of a bidisperse suspension of bubbles rising in a vertical channel , JUAN CARLOS SERRANO, SANTOS MENDEZ, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — Experiments were performed in a vertical channel to study the behaviour of a bidisperse suspension of bubbles. Bubbles were produced using capillaries of two distinct inner diameters. The capillaries are small enough to generate bubbles in the range of 1 to 6 mm in diameter. Using water and water-glycerin mixtures, the vertical component of the fluctuating liquid velocity was obtained using a flying hot wire anemometer technique. The system is characterized by the dimensionless Reynolds and Weber numbers in the range of $22 < Re < 890$ and $0.2 < We < 2.9$, respectively. The gas volume fraction ranges between 0.5% and 6%. The measurements were obtained for three ratios R of gas volume fraction: R = 1/3, 1 and 3, where $R = \alpha_{large}/\alpha_{small}$. The results show that the flow agitation, characterized by the liquid velocity variance increases with bubble concentration. We also found that the variance, normalized with the mean bubble velocity squared, $T_f = U_f'^2/U_b^2$, increased as the Reynolds number decreased. Bidisperse flows, in general, show larger values of fluctuation.

9:31AM AK.00008 Determination of Hydrodynamic Parameters on Two–Phase Flow Gas - Liquid in Pipes with Different Inclination Angles Using Image Processing Algorithm , GUSTAVO MONTOYA, MARÍA VALECILLOS, CARLOS ROMERO, DOSINDA GONZÁLES — In the present research a digital image processing-based automated algorithm was developed in order to determine the phase's height, hold up, and statistical distribution of the drop size in a two-phase system water-air using pipes with 0°, 10°, and 90° of inclination. Digital images were acquired with a high speed camera (up to 4500fps), using an equipment that consist of a system with three acrylic pipes with diameters of 1.905, 3.175, and 4.445 cm. Each pipe is arranged in two sections of 8 m of length. Various flow patterns were visualized for different superficial velocities of water and air. Finally, using the image processing program designed in Matlab/Simulink®, the captured images were processed to establish the parameters previously mentioned. The image processing algorithm is based in the frequency domain analysis of the source pictures, which allows to find the phase as the edge between the water and air, through a Sobel filter that extracts the high frequency components of the image. The drop size was found using the calculation of the Feret diameter. Three flow patterns were observed: Annular, ST, and ST&MI.

9:44AM AK.00009 Bubble Size Control Mechanisms and Effect on Flow Regime , THOMAS SHEPARD, University of Minnesota, PAUL STRYKOWSKI — Research has been conducted in an effort to understand the impacts of different control mechanisms on bubble size during air injection into a liquid cross-flow. The motivation for this work is to gain better control during the bubble formation and coalescence processes in order to reliably generate bubbles of different sizes. In this study air is injected through a porous plate into an electrolyte solution flowing through an adjustable geometry channel. The control mechanisms considered include the pore size in the porous plate, the channel height (and thus shear rate) at the injection site, and the electrolyte concentration. The effects of the controls are studied for a range of channel pressures (10-60 psi) and gas to liquid mass flow-rate ratios (0.001-0.005). By varying the different controls, bubbles are generated with a mean diameter of 80-1600 microns while keeping the pressure and gas to liquid mass flow-rate constant. An additional outcome of the research is the demonstrated effect that bubble size has on the transition from bubbly flow to slug flow.

9:57AM AK.00010 Disturbances to Air-Layer Skin-Friction Drag Reduction at High Reynolds Numbers[1] , DAVID DOWLING, BRIAN ELBING, SIMO MAKIHARJU, ANDREW WIGGINS, MARC PERLIN, STEVEN CECCIO, University of Michigan — Skin friction drag on a flat surface may be reduced by more than 80% when a layer of air separates the surface from a flowing liquid compared to when such an air layer is absent. Past large-scale experiments utilizing the US Navy's Large Cavitation Channel and a flat-plate test model 3 m wide and 12.9 m long have demonstrated air layer drag reduction (ALDR) on both smooth and rough surfaces at water flow speeds sufficient to reach downstream-distance-based Reynolds numbers exceeding 100 million. For these experiments, the incoming flow conditions, surface orientation, air injection geometry, and buoyancy forces all favored air layer formation. The results presented here extend this prior work to include the effects that vortex generators and free stream flow unsteadiness have on ALDR to assess its robustness for application to ocean-going ships. Measurements include skin friction, static pressure, airflow rate, video of the flow field downstream of the injector, and profiles of the flowing air-water mixture when the injected air forms bubbles, when it is in transition to an air layer, and when the air layer is fully formed. From these, and the prior measurements, ALDR's viability for full-scale applications is assessed.

[1]Sponsored by the Office of Naval Research.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AL CFD I: Methods 200A

8:00AM AL.00001 A divergence preserving Adaptive Mesh Refinement strategy for viscous incompressible flows[1] , M. VANELLA, E. BALARAS, University of Maryland — Structured adaptive mesh refinement (S-AMR) concentrates computational resources (i.e. grid points) in high-gradient regions of the flow, while maintaining most of the desirable properties of structured Cartesian solvers. Whenever the computational grid is locally refined/derefined the flow variables in S-AMR calculations need to maintain certain conservation properties during restriction or prolongation operations. Restriction refers to the transfer of a flow variable from a grid at a fine level of refinement to an underlying grid at a coarser level, while prolongation is the data transfer in the inverse direction. Of particular interest in S- AMR applications in viscous incompressible flows are divergence-preserving prolongation operators of a vector field. When the mesh refinement-derefinement procedure is applied after the predictor step of the fractional step integration scheme, divergence preservation for prolongation is crucial to avoid spurious pressure oscillations and additional errors on the computed flow field. In this work we propose method for divergence-preserving prolongation applicable to nested grids that differ by a factor of two in terms of resolution. The accuracy of the method is evaluated on prototypical laminar flows, like the Taylor-Green vortex problem and flow around a cylinder.

[1]Research supported by the National Science Foundation and AFOSR.

8:13AM AL.00002 Towards a Fully Adaptive Mesh-Free Method for Solving Viscous Incompressible Flows , PARITOSH MOKHASI, DIETMAR REMPFER, IIT, Chicago — A fully adaptive mesh-free method based on radial basis functions (RBF) is proposed for numerically solving the Navier- Stokes equations. The scheme is based on the method of lines wherein the spatial derivatives are approximated using a differential quadrature approach. The solution is progressed in time using a fractional step method with pressure correction. To demonstrate its flexibility, the 2D driven cavity problem is solved in the Eulerian and semi-Lagrangian framework using radial basis functions. We further demonstrate, via a 1D spatio- temporal example, that using RBFs adaptively enables one to produce highly accurate results. Finally, we present algorithms for solving a large class of fluid dynamics problems using radial basis functions.

8:26AM AL.00003 A spectral multidomain penalty method model for high Reynolds number incompressible flows , JORGE ESCOBAR-VARGAS, PETER DIAMESSIS, Cornell University — We present our latest results towards the development of a spectral multidomain penalty method-based incompressible Navier-Stokes solver for high Reynolds number stratified turbulent flows in doubly non-periodic domains. Temporal discretization of the governing equations is based on three fractional steps (explicit advancement of nonlinear terms and implicit treatment of pressure and viscous terms). The spatial discretization uses a Legendre collocation approach in discontinuous quadrilateral subdomains. Numerical stability is enabled through a penalty scheme, spectral filtering and appropriately defined dealiasing. The conditioning of the linear system associated with the discretized Poisson equation for the pressure is analyzed in detail. In addition, the efficiency of various preconditioning strategies such as diagonal and block Jacobi, finite difference, and additive Schwartz are investigated. Finally, the efficiency and accuracy of the Navier Stokes solver are assessed through application to select test cases.

8:39AM AL.00004 Wavelet regularization of the 2D incompressible Euler equations , ROMAIN NGUYEN VAN YEN, MARIE FARGE, LMD-CNRS, ENS, Paris, KAI SCHNEIDER, CMI, Universite de Provence — We examine the viscosity dependence of the solutions of two-dimensional Navier-Stokes equations in periodic and wall-bounded domains, for Reynolds numbers varying from 10^3 to 10^7. We compare the Navier-Stokes solutions to those of the regularized two-dimensional Euler equations. The regularization is performed by applying at each time step the wavelet-based CVS filter *(Farge et al., Phys. Fluids, 11, 1999)*, which splits turbulent fluctuations into coherent and incoherent contributions. We find that for Reynolds 10^5 the dissipation of coherent enstrophy tends to become independent of Reynolds, while the dissipation of total enstrophy 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. This indicates that a balance may have been established between coherent enstrophy dissipation and coherent enstrophy production at the wall. The Reynolds number for which the dissipation of coherent enstrophy becomes independent on the Reynolds number is proposed to define the onset of the fully-turbulent regime.

8:52AM AL.00005 A Multigrid Accelerated High-Order Compact Fractional Step Method for Unsteady Incompressible Viscous Flows , OMER SAN, ANNE STAPLES, Virginia Polytechnic Institute and State University — An efficient high-order compact scheme is presented for computing unsteady incompressible viscous flows. The scheme is constructed on a staggered Cartesian grid. Using the fractional step framework, the Navier-Stokes equations are advanced in time with the second-order Adams-Bashforth method without considering the pressure terms in the predictor step. The velocity field is then corrected so that the continuity equation is satisfied through a pressure Poisson equation. Since the efficiency of the fractional step method depends on the Poisson solver, a Mehrstellen-based V-cycle multigrid acceleration is implemented in the solution of the Poisson equation to enhance the computational efficiency. The method is validated by simulating a decaying Taylor-Green vortex. The results show that the method has high resolving efficiency, drastically reduced computational time, and high-order accuracy, making it applicable for the simulation of complex turbulent flows.

9:05AM AL.00006 Vortex-induced vibrations of a long flexible cylinder in transitional and turbulent flows , REMI BOURGUET, MIT, GEORGE KARNIADAKIS, Brown University, MICHAEL TRIANTAFYLLOU, MIT — The flow past a flexible cylinder subject to Vortex-Induced Vibrations (VIV) is investigated by direct numerical simulation at low and moderate Reynolds numbers (Re$= 100 - 1000$). The cylinder of large spanwise extension (≥ 200 diameters) is pinned and hinged at both ends and its central part is free to move in all directions under the effect of fluid-structure interaction. The cylinder dynamic is governed by a beam-cable equation. The influence of Reynolds number and structural parameters such as tension, bending stiffness and mass ratio, on VIV amplitudes and characteristic frequencies is quantified. The relationship between hydrodynamic efforts, structure motion and vortex shedding pattern is examined during the transition to turbulence. In particular, modifications of the alternating shedding pattern related with specific VIV conditions are analyzed in respect to the appearance of space/time irregularities in the structure response.

9:18AM AL.00007 A novel computational method to determine the dynamics of a lipid bilayer vesicle in a viscous flow[1] , DAVID SALAC, MICHAEL MIKSIS, Northwestern University — Models of lipid bilayer vesicle motion require that both the local area element of the interface and the volume enclosed by the interface be conserved. Here we present a novel level-set computational method to predict the dynamics of a vesicle under the influence of an external viscous fluid. The fluid both inside and outside the vesicle is governed by the Navier-Stokes equations. We impose both the volume and area constraint by implementing a novel splitting scheme. Similar to standard pressure-correction methods for the Navier-Stokes equations, which require the velocity field to be divergence free, we solve a variable coefficient pressure-Poisson equation with Neumann boundary conditions to ensure volume conservation. We also impose the constraint that the velocity field must be divergence free on the moving interface. This necessitates the solution of an additional partial differential equation. This equation and the needed boundary conditions will be presented. Numerical examples of the scheme and convergence checks will also be presented.

[1]Research supported by NSF Grant DMS-0636574.

9:31AM AL.00008 Two layer fluid stress analysis during airway closure , CHENG-FENG TAI, University of Michigan, DAVID HALPERN, University of Alabama, JAMES GROTBERG, University of Michigan — The airways are lined with a film consisting of two immiscible liquids, a serous layer and a more viscous mucus layer. Due to a surface tension driven instability, a liquid plug can form that obstructs the passage of air along the airways provided the ratio of the film thickness to the tube radius is greater than a critical value ∼0.12. In this study, we assume that the liquid layers are Newtonian, the surface tension is constant at the interfaces and the air-core phase is passive. We solve the Navier-Stokes and continuity equations subject to interfacial stress conditions and kinematic boundary conditions numerically using a finite volume approach in conjunction with a sharp interface method for the interfaces. Surface tension, viscosity and film thickness ratios can be altered by disease, and their influence on the closure instability is investigated. Results show that the shear and normal stresses along the airway walls can be strong enough to injure airway epithelial cells. We acknowledge support from the National Institutes of Health grant number NIH HL85156.

9:44AM AL.00009 Feasibility of using Hybrid Wavelet Collocation - Brinkman Penalization Method for Shape and Topology Optimization , 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 and topology optimization of fluid flows. Adaptive wavelet collocation method tackles the problem of efficiently resolving a fluid flow on a dynamically adaptive computational grid in complex geometries (where grid resolution varies both in space and time time), while Brinkman volume penalization allows easy variation of flow geometry without using body-fitted meshes by simply changing the shape of the penalization region. The use of Brinkman volume penalization approach allow seamless transition from shape to topology optimization by combining it with level set approach and increasing the size of the optimization space. The approach is demonstrated for shape optimization of a variety of fluid flows by optimizing single cost function (time averaged Drag coefficient) using covariance matrix adaptation (CMA) evolutionary algorithm.

9:57AM AL.00010 Uncertain shape optimization for dense gas flows , PIETRO MARCO CONGEDO, CHRISTOPHE CORRE, LEGI-Grenoble, JEAN-MARC MARTINEZ, CEA-Saclay — Uncertain shape optimization is a fascinating but challenging task. Our work explores some key issues in uncertain optimization and proposes a strategy to obtain a more reliable solution at a moderate computational cost. The steady transonic inviscid flow of a dense gas over an airfoil is considered and a shape optimization performed to minimize the airfoil's drag coefficient. Three sources of uncertainties are accounted for : thermodynamic model, freestream conditions and geometry. The combined effect of these uncertainties is analyzed to get the average and variance of the drag coefficient, that are both minimized during the optimization. Preliminary stochastic simulations based on polynomial chaos expansions yield the most influent uncertain parameters; several optimization strategies are then studied, with an original combination of response surfaces and metamodels, to obtain robust optimal solutions for a limited number of flow computations.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AM Reacting Flows I 200B

8:00AM AM.00001 Flamelet model for supersonic combustion[1] , VINCENT TERRAPON, HEINZ PITSCH, RENE PECNIK, Stanford University — The vast majority of computational work in supersonic turbulent combustion has so far relied on simplified/reduced mechanisms and the explicit transport of the involved species. Such approaches require then closure of the chemical source term in the species transport equation. An alternative approach is based on the flamelet concept which assumes that the chemical time scales are shorter than the turbulent time scales so that the flame can be approximated as one-dimensional. However, the implementation of the flamelet model is based on a low Mach number assumption, explaining the still very limited number of studies of high speed flows using this approach. Since supersonic speed and compressibility effects play an important role at supersonic speeds, the flamelet implementation has been reformulated where temperature is not any longer given by a chemistry table but computed from the total energy and the tabulated species mass fractions, thus, better accounting for compressibility effects. The model is applied to the combustor of the HyShot II vehicle and results are compared to experiment measurements and simulation data.

[1]This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number NA28614.

8:13AM AM.00002 LES/PDF Modeling of Soot Evolution in Turbulent Flames , VENKATRAMANAN RAMAN, The University of Texas at Austin, MICHAEL MUELLER, GUILLAUME BLANQUART, HEINZ PITSCH, Stanford University — Modeling soot evolution is turbulent flames is a complex problem due to the nonlinear interactions between the soot particles and the gas-phase turbulent combustion process. In this work, we develop a transported probability density function (PDF) approach for soot description in the context of large eddy simulation (LES) based combustion modeling. The soot number density is described using the bivariate VS (volume-surface) approach. The number density evolution equation is discretized using the hybrid method of moments technique, where the first four moments of the number density function are solved. In the transported PDF approach, the joint subfilter distribution of the gas-phase thermochemical scalars and the soot moments are evolved using a Lagrangian Monte-Carlo approach. The LES-PDF approach is validated using a piloted diffusion flame experiment. Results indicate that the PDF approach predicts delayed inception of soot particles and lower soot volume fraction as compared to the pure LES approach. While the soot volume fraction along the centerline of the jet is overpredicted by the simulation, the radial distribution is underpredicted as compared to the experiments. Further, the influence of mixing model on soot evolution is discussed.

8:26AM AM.00003 Simulation of an ethylene-air jet flame with soot and radiation modeling[1] , JEFFREY DOOM, JOSEPH OEFELEIN, Sandia National Laboratories — Large eddy simulation of an ethylene-air diffusion flame and supporting direct numerical simulations are presented. A reduced mechanism recently developed by Wang et al. is used (22 species, 107 reactions) and a systematic study is performed which compares the reduced mechanism to the original full mechanism (*USC Mech Version II*: 111 species, 784 reactions). A series of calculations are then validated by comparing results with CHEMKIN, Lignell et al. (*Combust. Flame 2007*) and the premixed experiments from Bhargava & Westmoreland (*Combust. Flame 1998*). The baseline soot model employed is from Leung et al (*Combust. Flame 1991*) and accounts for nucleation, growth, oxidation and coagulation. This model is coupled through source terms as a function of C_2H_2, CO, O_2 and H_2. The first two moments are considered to account for the number density and soot mass per volume. Initially the radiation model assumes an optically thin medium in a manner consistent with Lignell et al. Results associated with the soot model will be presented along with comparisons with experimental data.

[1]Supported by Strategic Environmental Research and Development Program.

8:39AM AM.00004 Direct Quadrature Method of Moments for LES-based Modeling of Supersonic Combustion , PRATIK DONDE, HEESEOK KOO, VENKAT RAMAN, University of Texas at Austin — The LES/transported probability density function (PDF) model has been successfully used for predictive modeling of turbulent combustion in low-speed flows. The PDF approach evolves the joint-distribution of the gas-phase thermochemical composition and is ideally suited for supersonic flows, where conserved-scalar approaches are not valid due to the compressible nature of the flow. In low-speed flows, the high-dimensionality of the PDF transport equation is handled through the use of Monte-Carlo based stochastic methods. However, the presence of shocks and large density and pressure gradients pose significant challenges in the use of these stochastic methods for high-speed flows. In this work, we propose a direct quadrature method of moments (DQMOM) approach, which is a fully Eulerian method for solving the PDF transport equation. Here, the subfilter PDF is discretized in terms of a finite number of delta functions, each characterized by a weight and an abscissa. Eulerian transport equations for these quantities are similar in structure to scalar transport equations and can be solved using finite-volume/finite difference approaches. Here, the accuracy of the DQMOM approach and the numerical implementation of this method using shock-capturing schemes are discussed.

8:52AM AM.00005 Effects of Acoustic Excitation on Bluff-body Stabilized Premixed Reacting Flows , VAIDYANATHAN SANKARAN, ROBERT ERICKSON, MARIOS SOTERIOU, United Technologies Research Center — Bluff body stabilized flames are used in numerous combustion applications to enable stable burning at high speeds. These confined flames are susceptible to acoustic excitation arising due to the confinement that can lead to thermoacoustic instabilities which are detrimental to the operability of the combustion device. In this study, we formulate a computational approach for the simulation of this phenomenon that is based on the one way coupling of an acoustic solution to a low Mach number but dilatational reacting flowfield. The latter is simulated with a purely Lagrangian and grid free approach that captures the rotational flowfield using the discrete Vortex Method and the reacting field by a kinematical solution of the G-equation. Earlier studies using this flow simulation approach have shown that the unsteady interactions, such as the transition from the asymmetric Von-Karman shedding to the more symmetric shedding structure present when reaction occurs can be captured accurately. Flame response to longitudinal acoustic waves is simulated and results are to be discussed in the context of transfer functions of heat release response to acoustic velocity excitation. Dominant mechanisms by which the flame responds to acoustics will also be identified. Finally, results are to be contrasted to those from analytical models that are in use in thermoacoustic studies today.

9:05AM AM.00006 The Interaction of High-Speed Turbulence with Flames[1] , ALEXEI POLUDNENKO, ELAINE ORAN, Naval Research Laboratory — Interaction of flames with turbulence occurs in systems ranging from chemical flames on Earth to thermonuclear burning fronts in supernovae. We present results of a systematic study of the dynamics and properties of turbulent flames formed under the action of high-speed turbulence in stoichiometric hydrogen-air mixture. Numerical simulations were performed using the massively parallel reactive-flow code Athena-RFX. Here we discuss (1) global properties of the turbulent flame in this regime (flame width, speed, etc.); (2) the internal structure of the flame brush; and (3) the internal structure of the flamelets folded inside the flame brush. We demonstrate that, in the case of hydrogen, turbulence does not affect the internal flame structure essentially for all subsonic turbulent intensities. We address the relative role of large-scale and small-scale motions on global and local properties of the turbulent flame. We also consider the processes that determine the turbulent burning velocity and identify two distinct regimes of flame evolution. Finally, we discuss the effects of non-equilibrium non-Kolmogorov turbulence on the turbulent flame properties.

[1]This work was supported in part by the National Research Council, Naval Research Laboratory, and the Office of Naval Research, and by the National Science Foundation through the TeraGrid resources.

9:18AM AM.00007 Transport by molecular diffusion in LES of a turbulent diffusion flame , KONSTANTIN KEMENOV, SHARADHA VISWANATHAN, HAIFENG WANG, STEPHEN POPE, Cornell University — Molecular diffusion effects in LES of a piloted methane-air (Sandia D) flame are investigated on a series of grids with progressively increased resolution towards the DNS limit. The role of molecular diffusivity in effecting spatial transport is studied by drawing a comparison with the turbulent diffusivity and analyzing their statistics conditioned on temperature. Statistical results demonstrate that the molecular diffusivity in the near-field almost always exceeds the turbulent diffusivity, except at low temperatures (less than 500K). Thus, by altering the jet near-field, molecular transport plays an important role in the further downstream jet development. Molecular diffusivity continues to dominate in the centerline region throughout the flow field. Overall, the results suggest the strong necessity to represent molecular transport accurately in LES studies of turbulent reacting flows.

9:31AM AM.00008 A LES/FDF/PMC-based Detailed Model for Luminous Turbulent Flames[1] , ANKUR GUPTA, Graduate Student, Dept of Mechanical and Nuclear Engg, The Pennsylvania State University, DANIEL HAWORTH, MICHAEL MODEST, Professor, Dept of Mechanical and Nuclear Engg, The Pennsylvania State University — A comprehensive model is presented for luminous turbulent flames with full consideration of turbulence/chemistry interactions (TCI), turbulence/radiation interactions (TRI), and detailed gas- phase chemistry and soot. A large-eddy simulation/composition filtered mass density function (LES/C-FDF) formulation is adopted that accounts exactly for the influence of subfilter-scale turbulent fluctuations on chemical source terms and radiative emission. A consistent Lagrangian Monte Carlo particle/Eulerian mesh method is used to solve the modeled C-FDF equation. A second Monte Carlo particle method (photon Monte Carlo - PMC) is used to solve the radiative transfer equation; the radiation model includes spectral radiation properties and absorption. Soot is modeled using a method of moments. The model is validated using experimental data for luminous turbulent nonpremixed jet flames. The model then is exercised to isolate and quantity different contributions to TRI.

[1]NSF Grant: CTS-0121573; NASA Grant: NNX07AB40A

9:44AM AM.00009 Flow field structure near the reaction zone in turbulent nonpremixed jet flames , MIRKO GAMBA, Stanford University, NOEL T. CLEMENS, OFODIKE A. EZEKOYE, The University of Texas at Austin — Quasi-instantaneous pseudo-volumes of the 3D velocity field in the far field of turbulent nonpremixed jet flames are constructed from cinematographic kilohertz-rate stereoscopic PIV applying Taylor's hypothesis. Jet flames at jet exit Reynolds numbers of 8,000-15,000 were considered. The approach enable computation of all nine velocity gradients and the 3D kinematic quantities. 10 Hz OH PLIF imaging was also included to mark the reaction zone. Three-dimensional rendering of regions of intense vorticity and energy dissipation reveals their sheet-like nature and their tendency to exist near the OH layers. Contrary to nonreacting jets, this feature is believed to be a due to the stabilizing effect of heat release and the laminar shear caused by the flame. Single-point statistics of the velocity gradients indicate anisotropy in the flow with strong gradients predominantly in the radial direction. However, the 1D energy spectrum and single-point statistics of the principal strain and strain-vorticity alignment follow the known trends from incompressible turbulence.

9:57AM AM.00010 Effects of Swirl on Strongly-Pulsed Turbulent Diffusion Flames , Y.-H. LIAO, J.C. HERMANSON, University of Washington — The dynamics of large-scale structures in strongly-pulsed, swirling, turbulent jet diffusion flames were examined experimentally. The combustor used a combination of axial and tangentially-injected air to produce a range of swirl numbers. Gaseous ethylene fuel was injected through a 2 mm diameter nozzle on the combustor centerline with a jet-on Reynolds number of 5000. The flames were fully-modulated, with the fuel flow completely shut off between pulses. High-speed imaging of the flame luminosity was employed to examine the flame dimensions and the celerity of the large-scale flame structures. The flames were found to be approximately 15-20% shorter when swirl was imposed, depending on the injection time. The more compact flames in swirl appear to be due to the presence of recirculation inside the flames. For longer injection times, the celerity of the flame structures generally decreases as the swirl intensity increases. This is evidently due to the reversed velocity in the recirculation zone. For shorter injection times, the flame celerity has an increasing trend with increased swirl intensity due to flames being closer to the fuel nozzle at burnout.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AN Experimental Techniques I 200C

8:00AM AN.00001 Evaluation of burst-mode LDA spectra with implications , CLARA VELTE, DTU, WILLIAM GEORGE, Chalmers University of Technology — Burst-mode LDA spectra, as described in [1], are compared to spectra obtained from corresponding HWA measurements using the FFT in a round jet and cylinder wake experiment. The phrase "burst-mode LDA" refers to an LDA which operates with at most one particle present in the measuring volume at a time. Due to the random sampling and velocity bias of the LDA signal, the Direct Fourier Transform with accompanying weighting by the measured residence times was applied to obtain a correct interpretation of the spectral estimate. Further, the self-noise was removed as described in [2]. In addition, resulting spectra from common interpolation and uniform resampling techniques are compared to the above mentioned estimates. The burst-mode LDA spectra are seen to concur well with the HWA spectra up to the emergence of the noise floor, caused mainly by the intermittency of the LDA signal. The interpolated and resampled counterparts yield unphysical spectra, which are buried in frequency dependent noise and step noise, except at very high LDA data rates where they perform well up to a limited frequency.

[1] Buchhave, P. **PhD Thesis**, SUNY/Buffalo, 1979.
[2] Velte, C.M. **PhD Thesis**, DTU/Copenhagen, 2009.

8:13AM AN.00002 Two-dimensional velocity measurements using Laser-Cantilever-Anemometry in comparison to x-wire anemometry , MICHAEL HÖLLING, JAROSLAW PUCZYLOWSKI, JOACHIM PEINKE, ForWind, Institue of Physics - University of Oldenburg — We present an improved 2D Laser-Cantilever-Anemometer (2D LCA) which allows for measurements in two dimensions. The two velocity components are resolved by detecting the bending and the torsion of a tiny cantilever using the laser pointer principle. Thereby a two dimensional position sensitive detector measures the movement of the reflected laser light coming from the cantilever. Measurements carried out with this 2D LCA in comparison to x-wire data acquired in the wake of a cylinder are presented.

8:26AM AN.00003 ABSTRACT WITHDRAWN –

8:39AM AN.00004 Design and Characterization of an Optical Feedback-Controlled Microphone for Aeroacoustics Research[1] , ELIOTT RADCLIFFE, AHMED NAGUIB, Michigan State University, MI-48824, WILLIAM HUMPHREYS, JR., NASA Langley Research Center, VA-23062 — An optical feedback-controlled microphone was designed and tested for potential use in phased "beam-forming" arrays used in aeroacoustics research. Optical sensing was employed as a means for measuring center displacement of a stretched thin membrane due to incident acoustic pressure. The membrane was constructed of PVDF which exhibits piezoelectric properties allowing actuation of the membrane in a feedback system. The latter was used to actively modify sensor parameters, most notably membrane stiffness, resonant frequency, and damping. Testing of a prototype microphone was performed using a plane wave tube calibrator. The results demonstrate that feedback control is an effective method for improving the microphone's transient response, as well as for "self-tuning" and matching of microphone parameters in sensing arrays.

[1]Supported by NASA GSRP grant NNX08AO07H.

8:52AM AN.00005 Measurements of instantaneous temperature in oscillating flows[1] , PHILIPPE BLANC-BENON, CNRS, ARGANTHAEL BERSON, Queen's University, GAELLE POIGNAND, GENEVIEVE COMTE-BELLOT, Ecole Centrale de Lyon, ECOLE CENTRALE DE LYON, FRANCE COLLABORATION, QUEEN'S UNIVERSITY, CANADA COLLABORATION — Temperature fluctuations in turbulent flows are usually investigated using cold wires operated by a constant-current anemometer. However, the output voltage of such anemometers is not hardware compensated for the thermal inertia of the wire. A correction is applied only during the post-processing of the data and requires the knowledge of the time lag of the wire, which depends both on the wire properties and on the instantaneous incident flow velocity. Here, a simple procedure for the instantaneous correction of the thermal inertia of cold wires is proposed. The method relies on the splitting of the time lag of cold wires operated in a constant-current mode into two factors: one depending on the wire properties and the other depending on flow velocity. These two factors are obtained from the operation of the wire by a constant-voltage anemometer in the heated-mode. The uncompensated signal delivered by the constant-current anemometer operating the cold wire is then processed to restore the signal that would be delivered by an ideal cold wire. Validation experiments are conducted in an acoustic standing-wave resonator where large-amplitude oscillatory flows take place.

[1]Supported by ANR MicroThermAc NT051_42101.

9:05AM AN.00006 Use of grid generated turbulence to assess hot-wire spatial resolution and Pitot probe turbulence corrections , ANAND ASHOK, MOHAMMAD JAVED, SEAN BAILEY, ALEXANDER SMITS, Princeton University — The objective of the present study is to use grid generated homogeneous isotropic turbulence as a benchmark flow to test the effect of turbulence on different measurement techniques. The grid turbulence is generated in a low speed 2 foot by 3 foot closed circuit wind tunnel using a 1 inch square mesh grid placed at the test section inlet. Measurements of the turbulence using a variety of hot-wire sensor lengths, at a series of streamwise distances downstream of the grid, will be used to investigate the impact of spatial filtering on different turbulence statistics and fully characterize the grid turbulence. Points of comparison will include basic mean and turbulent kinetic energy profiles as well as higher order statistics, the turbulent dissipation rate and finally turbulent spectra. These statistics will then be combined with additional measurements to assess the performance of turbulence corrections for Pitot probe velocity measurements. Finally, it is hoped that this flow can be used to assess the performance of a new nano-scale hot wire probe currently under development.

9:18AM AN.00007 The accuracy of cross-stream velocity gradients measured by multi-sensor hot-wire probes , MILAN ŠEKULARAC, PETAR VUKOSLAVČEVIĆ, University of Montenegro, JAMES WALLACE, ELIAS BALARAS, NIKOLAOS BERATLIS, University of Maryland — A highly resolved turbulent minimum channel flow DNS with $Re_\tau = 180$ was used to investigate the effects on the accuracy of simultaneous measurements of velocity gradient components resulting from the spatial resolution and sensor arrangement of twelve-sensor hot-wire probes. The sensors were represented as points on the simulation grid, the effective velocity cooling each sensor was determined and sensor equations were then solved in response to the DNS field to obtain velocity and velocity gradient components. It has been found that all cross-stream velocity gradients except $\partial v/\partial y$ and $\partial w/\partial z$ can be measured with reasonable accuracy. Depending on the arrangement of the sensors and the array and probe sizes, either one or the other of these two gradients is subject to high measurement error in the near wall region of this bounded flow. As a consequence, the estimation of $\partial u/\partial x$ from the direct measurement of $\partial v/\partial y$ and $\partial w/\partial z$ by applying the continuity equation for incompressible flow is questionable. It appears that this is the likely explanation for the weak correlation in the near wall region of $\partial u/\partial x$, estimated by applying Taylor's hypothesis, with its value estimated using the continuity equation as has been found in several investigations using twelve-sensor probes.

9:31AM AN.00008 Effect of Wind Tunnel Wall Adaptation on Flow over a Circular Cylinder , SERHIY YARUSEVYCH, MICHAEL BISHOP, University of Waterloo — The presence of test section walls in many experimental facilities give rise to blockage effects, which detrimentally influence experimental data. A unique method to eliminate such blockage effects is to adapt the walls of a test section so as to mimic the conditions of an unbounded flow. The effect of such wall adaptation on flow development over a circular cylinder was the focus of this investigation. Velocity and surface pressure measurements were made in three test section wall configurations: geometrically straight walls (GSW), aerodynamically straight walls (ASW), and streamlined walls (SLW). In all the wall configurations investigated, tests were conducted for $Re_d = 58{,}000$ and model blockage ratios of up to 17%. The results show that, in GSW and ASW, blockage effects significantly alter flow development, affecting separated shear layer instability frequency, vortex shedding frequency, and limiting wake growth. Streamlining the walls successfully mitigates these adverse effects, with the relevant flow parameters shown to match those obtained in previous investigations conducted at low blockage ratios. Although the blockage effects produce an increase of both the separated shear layer instability and the wake vortex shedding frequency in GSW and ASW, the results suggest the ratio of these frequencies is invariant with the wall configuration. A comparative analysis of experimental data is performed to explain the observed trends.

9:44AM AN.00009 Design of a High Viscosity Couette Flow Facility for Patterned Surface Drag Measurements[1] , TYLER JOHNSON, AMY LANG, University of Alabama — Direct drag measurements can be difficult to obtain with low viscosity fluids such as air or water. In this facility, mineral oil is used as the working fluid to increase the shear stress across the surface of experimental models. A mounted conveyor creates a flow within a plexiglass tank. The experimental model of a flat or patterned surface is suspended above a moving belt. Within the gap between the model and moving belt a Couette flow with a linear velocity profile is created. PIV measurements are used to determine the exact velocities and the Reynolds numbers for each experiment. The model is suspended by bars that connect to the pillow block housing of each bearing. Drag is measured by a force gauge connected to linear roller bearings that slide along steel rods. The patterned surfaces, initially consisting of 2-D cavities, are embedded in a plexiglass plate so as to keep the total surface area constant for each experiment. First, the drag across a flat plate is measured and compared to theoretical values for laminar Couette flow. The drag for patterned surfaces is then measured and compared to a flat plate.

[1]Funding for this research has been received from the Lindbergh Foundation and NASA AL-EPSCoR.

9:57AM AN.00010 Redesign of contraction, test section and diffuser for a six-inch high speed water tunnel , IVAYLO NEDYALKOV, MARTIN WOSNIK, University of New Hampshire — The six-inch high speed water tunnel was recently moved from St. Anthony Falls Laboratory to the University of New Hampshire, where it is being restored. This water tunnel was a 1:6 scale model for the 36-inch Variable Pressure Cavitation Tunnel at David Taylor Model Basin and was used in many fundamental cavitation studies in the past, including the development of Schiebe bodies. It originally had a 6-inch circular test section and was later retrofitted with a 7-inch octagonal test section. In order to increase the maximum achievable velocity in the test section and improve the flow quality a new 6-inch square test section with diminishing 1-inch fillets was designed, which also required the design of a new contraction and diffuser. Contraction, test section and diffuser configurations were studied parametrically using CFD. The numerical predictions are compared to results in the literature and measurements in the tunnel. Further improvements include a new motor and control system. The renovated six-inch tunnel will be used for research on control of cavitating flows, hydrofoil development and general cavitation studies.

Sunday, November 22, 2009 8:00AM - 10:10AM –

Session AP Instability: Boundary Layers I 200D

8:00AM AP.00001 Global optimal disturbances using time-steppers , ANTONIOS MONOKROUSOS, LUCA BRANDT, DAN S. HENNINGSON, Linne Flow Centre, KTH Mechanics — The global linear stability of boundary-layer flows subject to three-dimensional disturbances is studied by means of Lagrange optimization. We consider the optimal initial condition leading to the largest growth at finite times and the optimal harmonic forcing leading to the largest asymptotic response (pseudo-spectrum). Both optimization problems are solved using a Lagrange multiplier technique, where the objective function is the kinetic energy of the flow perturbations and the constraints involve the linearised Navier-Stokes. Whereas the computation of optimal initial condition is known in the time-stepper context, the formulation of the optimal forcing problem is novel. The approach proposed here is particularly suited to examine convectively unstable flows, where single global eigenmodes of the system do not capture the downstream growth of the disturbances. For spanwise wavelengths of the order of the boundary layer thickness finite-length streamwise vortices exploit the lift-up mechanism to create streaks. For long spanwise wavelengths the Orr mechanism combined with the amplification of oblique wave packets are responsible for the disturbance growth. The latter mechanism is found to be dominant for the relatively long computational domain and high Reynolds number considered here. The use of matrix-free methods enables us to extend the present framework to any geometrical configuration.

8:13AM AP.00002 Receptivity of Görtler Flow , LARS-UVE SCHRADER, LUCA BRANDT, DAN HENNINGSON, KTH Mechanics, Stockholm, Sweden, TAMER ZAKI, Imperial College London, UK — The flow over a concave surface, e.g. the lower side of a turbine blade, is subject to centrifugal forces which may destabilize the boundary layer. The instabilities appear as streamwise aligned counter-rotating vortices and may be steady or traveling, depending on the perturbation source. We consider the boundary layer on a concave wall with constant radius of curvature and expose the flow to two different disturbance sources: streamwise-localized, spanwise-sinusoidal roughness elements and free-stream vortical disturbances modeled by continuous-spectrum modes for the Blasius inflow. Results from numerical simulations using the Spectral Element Method (SEM) will be shown. The SEM provides spectral accuracy while allowing for geometries beyond the scope of global spectral methods based on Fourier modes. Owing to the non-parallel nature of the Görtler vortices, three-dimensional simulations are in particular appropriate to characterize the receptivity of the Görtler boundary layer.

8:26AM AP.00003 Boundary-layer transition in the wake of surface irregularities , JEFFREY CROUCH, The Boeing Company, VLADIMIR KOSORYGIN, Khristianovich ITAM, SB RAS, LIAN NG, The Boeing Company — Aerodynamic surfaces designed for laminar flow inevitably have geometric imperfections. These imperfections impact the unsteady processes in the boundary layer and may accelerate the laminar-turbulent transition. An experimental study is conducted to investigate the steady and unsteady disturbances in the wake of protruding and recessed surface irregularities, and to link these disturbances to the initial movement of the transition location. The steady disturbance field on the centerline of the irregularity is characterized by a region of velocity deficit followed by a much longer region of velocity surplus. Unsteady disturbances in the wake of the irregularity, measured prior to transition, have increased magnitudes and display a shift toward higher frequency. Local stability analysis is shown to capture many of the features of the pre-transitional flow. The transition Reynolds numbers collapse reasonably well when plotted in terms of the roughness height (non-dimensionalized by the boundary-layer displacement thickness). The initial movement of the transition can be represented by a reduction in the critical N-factor, consistent with a linear-amplitude based transition criterion.

8:39AM AP.00004 Calculating boundary layer receptivity to transiently growing roughness-induced perturbations using experimental data[1] , NICHOLAS DENISSEN, EDWARD WHITE, Texas A&M University — The receptivity problem is of great interest in perturbations generated by surface roughness. To quantify non-modal receptivity, continuous spectrum amplitude distributions are calculated for transiently growing roughness-induced perturbations in a flat-plate boundary layer. Complex, realistic, surface roughness is beyond the scope of direct numerical simulation (DNS) currently. This makes analyzing the receptivity of experimental results essential. A method using regularizing functionals is shown for calculating the distributions when only partial experimental data is available. These results are validated against DNS results. These amplitude distributions provide a way of rigorously characterizing the boundary layer receptivity to surface roughness. Extracting the continuous spectrum amplitudes using the partial data technique reveals the underlying vortex behavior that creates transient growth that is too difficult to measure experimentally. The method described is amenable to future work with realistic distributed roughness and complex surface geometries, and is applied to cases currently beyond the scope of DNS.

[1]This work is supported by the NSF and AFOSR.

8:52AM AP.00005 Control of Stationary Crossflow Modes in a Supersonic Boundary Layer using Distributed Roughness[1] , CHAN-YONG SCHUELE, ERIC MATLIS, THOMAS CORKE, University of Notre Dame, STEPHEN WILKINSON, P. BALAKUMAR, LEWIS OWENS, NASA Langley Research Center — Passive methods like distributed micron sized roughness elements have proven to work efficiently as subsonic laminar flow control devices. Attempts to experimentally extend the principle of suppression of the most amplified stationary cross flow modes to supersonic boundary layers have not been successful until now. This study presents evidence for the receptivity of a supersonic boundary layer with transition dominated by stationary cross flow modes to patterned roughness with different wave numbers. Experiments have been performed at the Mach 3.5 NASA LaRC Supersonic Low Disturbance Tunnel on a $7\,\mathrm{deg}$ half angle sharp cone at $4.3\,\mathrm{deg}$ angle of attack and a unit Reynolds number of $2.5x10^5 in^{-1}$. Pitot tube pressure measurements as well as surface flow visualization were used to detect the occurence of stationary crossflow modes and transition. Based on these two measurement approaches we conclude that the stationary cross-flow mode was receptive to the passive patterned roughness, indicating that control of transition to turbulence in cross-flow dominated conditions should be possible.

[1]Supported under the NASA NRA NNX08AB22A.

9:05AM AP.00006 Effect of Surface Thermal Perturbations on Compressible Boundary Layer Stability[1] , CHRISTOPHER ALBA, DATTA GAITONDE, Air Force Research Laboratory — High-speed laminar-turbulent boundary layer transition is a critical issue for re-entry and sustained hypersonic cruise vehicles. Turbulent wall heating rates can increase several orders of magnitude compared to laminar rates and skin friction drag can become a major component of the overall drag. We analyze approaches to modulate transition by altering the stability features of the boundary layer through the use of thermal perturbations. To this end, high-fidelity numerical simulations to generate basic states for Mach 1.5 and Mach 5.6 flat plate boundary layers with and without thermal bumps. Linear Parabolized Stability Equations (PSE) are solved using the STABL software suite to establish the flow stability characteristics under baseline (no excitation), constant and pulsed bump cases for each freestream Mach number. The effects are described in terms of neutral curves showing amplification for various frequencies versus Reynolds number. The three-dimensional flow structure is also examined near the breakdown to turbulence flow region to gain insight into the final stages of transition.

[1]Computational resources were made available by the DoD Supercomputing Resource Center at the Air Force Research Laboratory.

9:18AM AP.00007 Experiments and NPSE of roughness receptivity in swept-wing boundary layers[1] , WILLIAM SARIC, MATTHEW WOODRUFF, HELEN REED, Texas A&M University — New data are presented on 3-D boundary-layer receptivity to roughness in low-disturbance environments. The measurements include infra-red thermography with calibrated and temperature-compensated hotfilms to study roughness-related issues of boundary-layer transition in flight. A swept-wing model is mounted on the wing of a Cessna O-2 aircraft where nonlinear parabolized stability equations (NPSE) correlate the stability measurements and transition locations. The laminarization scheme of spanwise-periodic discrete roughness elements (DRE) is investigated at chord Reynolds numbers of 7.5 million. Flight experiments were conducted where the surface roughness amplitude was varied from 6 to 50 microns while the disturbance shear-stress was measured with calibrated hotfilm gauges in two locations: x/c= 15% and 30%; the former in the linear range and the later in the nonlinear range. In this way, the disturbance velocity amplitude was calculated as a function of roughness Reynolds number. These data were then used as initial conditions for the NPSE calculations to determine the efficacy of the DREs. The work was supported by: AFOSR Grant FA9550-05-0044, AFRL, and NASA Langley Research Center.

[1]The work was supported by: AFOSR Grant FA9550-05-0044, AFRL, and NASA Langley Research Center.

9:31AM AP.00008 Uncertainty quantification of the instability in a supersonic boundary layer with roughness , OLAF MARXEN, GIANLUCA IACCARINO, ERIC SHAQFEH, Stanford University — Knowledge of the location of laminar-turbulent transition on the surface of vehicles (re-)entering a planetary atmosphere is important for heat-shield design. However, due to the heat-shield material itself or as a result of ablation during flight, the surface of a heat shield is often not smooth. Instead, surface roughness occurs, but the height of this roughness may not be known beforehand. A numerical investigation of disturbance amplification in a laminar compressible flat-plate boundary layer with a localized 2-D roughness is carried out. Both linear and weakly non-linear disturbance evolution are considered. The non-linear case exhibits a secondary subharmonic resonance. In addition to deterministic simulations, a stochastic approach is applied to quantify uncertainties. The random parameter is chosen to be the height of the roughness in the linear case, while in the non-linear case the amplitude of the primary disturbance is considered a random parameter. Deterministic simulations show that the 2-D roughness acts as an amplifier for convective disturbances, and the resulting increased disturbance amplitude can enhance a secondary instability. The stochastic approach allows to quantify the probability for an increased or decreased amplification.

9:44AM AP.00009 ABSTRACT WITHDRAWN –

9:57AM AP.00010 Classification of the Flow Produced by an Oscillating Fence in a Laminar Boundary Layer , MICHAEL HIND, WILLIAM LINDBERG, JONATHAN NAUGHTON, University of Wyoming — Flow visualization has revealed that an oscillating fence produces a range of vortical structures in a flat plate laminar boundary layer. The structure can be classified by the ratio ϕ_0 of the fence oscillation frequency to the fundamental shedding frequency of the static fence. Particle image velocimetry was used to quantitatively investigate the flow structures of each classification regime. Fences operating in the subcritical flow regime ($\phi_0 < 0.1$) shed vortices due to vortex saturation behind the fence. The vortices of the critical flow regime ($\phi_0 \sim 1$) strengthen during the fence upstroke and are forced to shed once the fence begins to descend. The vortices of the supercritical flow regime ($\phi_0 > 1$) are shed once per fence oscillation cycle and coalesce to form larger vortices at the fundamental shedding frequency of the static fence. For the transitional flow regime ($\phi_0 \sim 0.1 - 1$), the structures are two-dimensional during the fence upstroke that become three-dimensional once the fence begins to descend. Through this classification system, it is possible to determine the frequency required for a given flow to produce the desired type of structure. By varying the fence frequency, the structure can be made to change dramatically.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AQ Instability: Interfacial and Thin-Film I 200E

8:00AM AQ.00001 Surfactant- and elasticity-induced inertialess instabilities in vertically vibrated liquids , SATISH KUMAR, BALRAM SUMAN, University of Minnesota — We investigate instabilities that arise when the free surface of a liquid covered with an insoluble surfactant is vertically vibrated and inertial effects are negligible. In the absence of surfactants, the inertialess Newtonian system is found to be stable, in contrast to the case where inertia is present. Linear stability analysis and Floquet theory are applied to calculate the critical vibration amplitude needed to excite the instability, and the corresponding wavenumber. A previously reported long-wavelength instability is found to persist to finite wavelengths, and the connection between the long-wavelength and finite-wavelength theories is explored in detail. The instability mechanism is also probed and requires the Marangoni flows to be sufficiently strong and in the proper phase with respect to the gravity modulation. For viscoelastic liquids, we find that instability can arise even in the absence of surfactants and inertia. Mathieu equations describing this are derived and these show that elasticity introduces an effective inertia into the system.

8:13AM AQ.00002 Anatomy of a wave , JÉRÔME HOEPFFNER, RALF BLUMENTHAL, STÉPHANE ZALESKI, Institut Jean le Rond D'Alembert, UMR 7190, Université Pierre et Marie Curie, Paris — A perturbation is induced at the sheared interface between a stream of liquid and a stream of gas. This initial perturbation then evolves as the response of inertia, viscosity and interfacial tension. We observe that the wave obtained by this procedure tends to a self-similar regime after a short transient. We describe the anatomy of this well-defined growing wave as the physical parameter are varied, in particular as the density ratio of the two phases is changed. This study is aimed at identifying a possible recurrent agent in atomization processes.

8:26AM AQ.00003 Interfacial flow control in two-phase systems with application to liquid bridges[1] , ILYA RYZHKOV, Institute of Computational Modelling SB RAS, VALENTINA SHEVTSOVA, Universite Libre de Bruxelles — We perform a theoretical study of thermocapillary flows and their stability in a two-phase system of infinite liquid column surrounded by the gas layer. This study is a complementary step in the JEREMI project (Japanese–European Research Experiment on Marangoni Instability). It is devoted to the development of efficient means for controlling thermocapillary flows in liquid bridges (columns) and scheduled to fly on ISS in 2011. The flows are controlled by applying mechanical stresses to the interface and varying the interfacial heat exchange by blowing gas around the liquid. The analytical solution describing stationary velocity and temperature profiles in the liquid and gas is derived. It is shown that liquid motion can be completely suppressed by the gas flow. The linear stability analysis of stationary flows is performed. It is shown that when the gas flow is opposite to (co-directed with) that of liquid on the interface, the system becomes more (less) stable. It occurs due to mechanical stresses applied to the interface and interfacial heat exchange. Consideration of liquid bridge with the surrounding gas provides better agreement with experimental results than previous calculations without gas phase.

[1]This work is supported by the RFBR Grant 08-01-00762a.

8:39AM AQ.00004 Stability and structure formation in films of binary mixtures[1] , SANTIAGO MADRUGA, Universidad Politecnica de Madrid, UWE THIELE, Loughborough University — Films of polymer blends are used in technological applications such as coatings or structured functional layers. The evolution of those films is involved by the coupling of decomposition within the film and the dewetting of the film. We present a model for films of binary mixtures, 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 analyze the linear stability of vertically stratified base states of free surface films with respect to lateral perturbations [1]. 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]. We study as well the dependence of the instability on parameters such as the Reynolds number, the surface tension number and the ratio of velocities of convective and diffusive transport. [1] U. Thiele, S. Madruga, and L. Frastia. Phys. of Fluids. 19, 122106, (2007). [2] S. Madruga and U. Thiele. To appear in Phys. of Fluids.

[1]S.M. acknowledges support via FP7 Marie Curie Reintegration Grant (PERG04-GA-2008-234384), by Universidad Politecnica de Madrid (AL09-P(I+D)), and U.T. by EU via FP7 (PITN-GA-2008-214919).

8:52AM AQ.00005 Dynamics and stability of turbulent falling films[1] , ALIKI MAVROMOUSTAKI, LENNON O'NARAIGH, OMAR MATAR, Imperial College London — We study the dynamics of thin turbulent films falling under the action of gravity. A base state, corresponding to a waveless film, is obtained by balancing gravity against viscous drag. The latter includes turbulent viscosity contributions characterised by a simple mixing length model. A linear stability analysis of this base state is then carried out leading to the derivation of an Orr-Sommerfeld-type eigenvalue problem. Numerical solutions of this problem reveal a Reynolds number-dependent competition between destabilising contributions arising from the turbulent base state and stabilising ones from the turbulent stresses at the interface and in the bulk. An energy budget analysis demonstrates clearly that the destabilising mode corresponds to an interfacial one. Our results also reveal that the most dangerous mode is in the long-wave regime. This provided motivation for the derivation of a long-wave model for the nonlinear film dynamics, which represents an extension of the Shkadov equations for turbulent falling films. The results of a brief parametric study of this model are presented.

[1]EPSRC Grant number EP/E021468/1

9:05AM AQ.00006 Coherent wave structures on falling fluid films flowing down a flexible wall , GRIGORI SISOEV, University of Birmingham, RICHARD CRASTER, University of Alberta, SATISH KUMAR, University of Minnesota, OMAR MATAR, Imperial College London — The dynamics of a thin fluid film flowing down a flexible vertical wall at moderate flow rates is studied in order to identify the dominant wave structures that will be observed in experiments. An asymptotic reduction using boundary-layer theory, and the von Kármán-Polhausen approximation, leads to coupled partial differential equations governing the nonlinear dynamics of the flow rate, and the gas-liquid and liquid-solid interfaces; closure is provided by a semi-parabolic fluid velocity profile. Fluid inertia, capillarity and viscous retardation effects are incorporated as are wall damping and tension. The validity of our approach is demonstrated using direct comparisons with predictions from the Orr-Sommerfeld equations. Nonlinear steady-travelling waves are identified from a nonlinear eigenvalue problem illustrating a multiplicity of solutions from which the dominating (attracting) solutions can be identified. Subsequent time-dependent numerical simulations of the fully-nonlinear partial differential equations demonstrate the selection of these dominant solutions, and, as such, they then constitute a point of direct comparison with physical experiments.

9:18AM AQ.00007 The Effect of Dynamic Wetting on the Stability of a Gas-Liquid Interface Subjected to Vertical Oscillations , ANDREW M. KRAYNIK, LOUIS ROMERO, JOHN R. TORCZYNSKI, CARLTON F. BROOKS, TIMOTHY J. O'HERN, RICHARD A. JEPSON, GILBERT L. BENAVIDES, Sandia National Laboratories — The stability of an interface in a container partially filled with silicone oil and subjected to gravity and vertical oscillations has been examined theoretically and computationally. An exact theory for the onset of a parametric instability producing Faraday-like waves was developed for arbitrary liquid viscosity, stress-free walls, and deep two-dimensional or axisymmetric containers. Finite-element simulations for stress-free walls are in excellent agreement with the theory, which predicts instability in discrete frequency bands. These simpler calculations are a departure point for examining the more realistic problem, which involves no-slip at the walls and dynamic wetting modeled with a Blake condition. 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.

9:31AM AQ.00008 On contact line induced instability in flow of hanging fluid films , TE-SHENG LIN, LOU KONDIC, New Jersey Institute of technology — We consider free surface instabilities of films hanging on inverted substrates within the framework of lubrication approximation. Contrary to all the previous works, we include fluid fronts in formulation. It is found that the presence of contact lines leads to free surface instabilities of convective type without any additional natural or excited perturbations. A single parameter $D = (3Ca)^{1/3} \cot \alpha$, where Ca is the capillary number and α is the inclination angle, is identified as a governing parameter in the problem. This parameter may be interpreted to reflect the combined effect of inclination angle, film thickness, Reynolds number and the fluid flux. Variation of D leads to change of the wave-like properties of the instabilities, allowing to observe traveling wave behavior, mixed waves, and the waves resembling solitary ones, which were observed in many other unstable flows.

9:44AM AQ.00009 Draw Resonance in Viscous Sheets , OLUS BORATAV, ZHEMING ZHENG, ALEXEY AMOSOV, Corning Incorporated — The instability known as the "draw resonance" in literature is studied for a viscous sheet considering the visco-gravity balances (Stokes number) and the heating/cooling effects (Stanton number). The analysis considers lubrication approximation for continuity, momentum and energy equations and determines the critical draw ratio for a range of Stokes numbers and Stanton numbers. The critical draw ratio is very sensitive to the variation of Stokes and Stanton numbers. It is shown that the decrease in Stokes number and/or the increase in Stanton number results in a decrease in the critical draw ratio.

9:57AM AQ.00010 Transience to Instabilty in a Liquid Sheet , NATHANIEL BARLOW, S.P. LIN, BRIAN HELENBROOK, Clarkson University — Series solutions are found which describe the evolution to absolute and convective instability in an inviscid liquid sheet flowing in an ambient gas and subject to a localized perturbation. These solutions are used to validate spatio-temporal stability predictions for sinuous and varicose modes. We show how recent disagreements in growth predictions stem from assumptions made when arriving at the Fourier integral response. Certain initial conditions eliminate (or reduce the order of) singularities in the Fourier integral. For the sinuous mode, deLuca and Costa (1997) predicted that an impulsive disturbance spreads both upstream and downstream and grows like $t^{1/3}$ when the Weber number is smaller than one. If a Gaussian perturbation is applied to both the position and velocity of the sheet, we observe this behavior in our series solution. However, when the initial disturbance velocity is taken to be zero, we find that the origin decays like $t^{-2/3}$. This is the growth predicted by Luchini (2004).

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AR Buoyancy Driven Flows 200F

8:00AM AR.00001 Icicles: diffusive gravity currents and phase change , M.G. WORSTER, J.A. NEUFELD, R.E. GOLDSTEIN, DAMTP, University of Cambridge — The growth and melting of icicles motivates this study of diffusive gravity currents. For example, the growth of an icicle vertically downwards is mediated by convective heat transfer in the surrounding air. The flow of the buoyant air is traditionally viewed as a near-vertical boundary layer and solved using the associated partial differential boundary-layer equations. However, near the tip of the icicle gravity acts in a direction orthogonal to the primary direction of flow. This situation is exemplified by the case of buoyancy-driven flow above a cooled, finite, horizontal plate or below a heated, finite horizontal plate. We find solutions of the corresponding boundary-layer equations in the form of approximate, nonlinear, separable solutions, with the horizontal variation of the boundary-layer thickness being governed by equations similar to those for a gravity current. We exploit the structure of such solutions to compute the steady-state shape and rate of melting of an icicle.

8:13AM AR.00002 Mean flow influence on a gravity current in a nearly horizontal confined geometry , THOMAS SEON, MOHAMMAD TAGHAVI, KRISTA THIELMANN, MARK MARTINEZ, IAN FRIGAARD, University of British Columbia — We study experimentally the effect of an imposed mean flow on a buoyancy-driven exchange flow of two miscible fluids of the same viscosity in a long tube, oriented close to horizontal. Measuring the front velocity, V_f, as a function of the mean flow velocity, V_0, for different density contrasts, viscosity, and inclination angles has allowed us to identify two regimes. First, for low V_0 the flow is dominated by the buoyancy-driven flow and the dynamics are similar to the exchange flow dynamics. Secondly, for high V_0 the flow is dominated by the imposed mean flow, the front velocity varies proportionally to the mean flow velocity and the ratio V_f/V_0 does not appear to be very sensitive to the density contrast or viscosity, but does increase as the tube gets closer to vertical. In this regime we find that the flow becomes more laminar and stable, as V_0 increases. This appears counter intuitive, since more energy is being injected into the system through the mean flow.

8:26AM AR.00003 Modified thermal theory for gravity currents on sloping boundaries , ALBERT DAI, Tamkang University — In this study we generalize the thermal theory adopted in Beghin, Hopfinger, and Britter (J. Fluid Mech. Vol. 107, 1981, p. 407) in order to account for both entrainment and detrainment effects occurring in the motion of gravity currents. We observe that although the model of Beghin et al. (1981) qualitatively captures the acceleration and deceleration phases of gravity current motion, their pure entrainment model consistently underpredicts the gravity current velocity and the distance before the maximum velocity is reached. Their model, therefore, could easily overestimate the arrival time of a gravity current generated by an instantaneous buoyancy release. We find that the effect of detrainment is to increase the predicted velocity of gravity current and extend the predicted distance before the maximum velocity is reached. The effect of detrainment is not immediately obvious, but it explains the differences between the experimental data and the model of Beghin et al. (1981). The idea presented here will lead to more investigations of gravity currents on sloping boundaries.

8:39AM AR.00004 Study of the Front Structures of Gravity Current Using Simultaneous PIV/PLIF Measurement , JUN CHEN, DUO XU, Purdue University, SCHOOL OF MECHANICAL ENGINEERING, PURDUE UNIVERSITY TEAM — The mixing and entrainment associated with the front development of gravity current have important implications in studying of many atmospheric and oceanic flow problems. A series of laboratory experiments are performed to investigate the development of the front structure of gravity current in an apparatus in which dense fluid is introduced into a less dense environment through a locking gate. A simultaneous PIV/PLIF system is developed to measure the velocity and density fields. The dynamics and structures around the current front are examined as well as the effect of bottom inclination.

8:52AM AR.00005 Jets, plumes and particle-laden jets in two-dimensional environments , J.R. LANDEL, C.P. CAULFIELD, BPI & DAMTP, University of Cambridge, A.W. WOODS, BPI, University of Cambridge — Results on the experimental investigation of liquid jets and plumes in a quasi Hele- Shaw cell are presented. Few experimental studies have been conducted when jets are constrained in a narrow gap whose length is two orders of magnitude smaller than the length scales of the other two dimensions. In this configuration, the dynamics shown by the jets is very rich when parameters such as the initial flow-rate and the buoyancy are changed. Furthermore, different behaviors have been observed for the front of the jet and the flow in steady state. In particular, the models for the rise of the jet and the expansion must be slightly modified between the two cases. PIV techniques have been used to measure accurately the flow field of the jets and to allow accurate comparison with the theoretical models. Finally, the results of this investigation on jets and plumes serve as a basis for more complex experiments involving particle- laden jets. A better understanding of liquid jets with varying buoyancy proves to be useful to the study of this two-phase flow.

9:05AM AR.00006 Instability Phenomena in Stratified, Particle-laden Flow , PETER BURNS, UC Santa Barbara, LUTZ LESSHAFFT, Ecole Polytechnique, ECKART MEIBURG, UC Santa Barbara — When a layer of particle-laden water is placed above clear water of different temperature and salinity, various instabilities may arise. Depending on the specific density configuration, distinct convection patterns ("fingering" vs. "leaking") have been reported from experiments (Parsons et al. 2001, Maxworthy 1999). We present linear stability results for such situations, with a focus on the role of particle settling. The effect of the settling velocity on the temporal instability growth rates is investigated in combination with various salinity distributions. The nonlinear evolution of the resulting instability structures is studied via DNS. Using linear analysis, DNS and experimental literature we hypothesize various mechanisms in an attempt to explain the occurrence of "leaking" and "fingering." Current nonlinear results will be presented in an effort to fully explain the "leaking" mode.

9:18AM AR.00007 Intrusion-generated internal waves in a constantly stratified fluid[1] , BENJAMIN MAURER, Scripps Institution of Oceanography, PAUL LINDEN, University of California, San Diego — Intrusive Gravity Currents (IGCs) occur when horizontal density gradients result in the intrusion of one fluid into another fluid at an intermediate depth. The vertical density stratification of the receiving fluid necessary for an IGC is also capable of supporting internal wave motion. Though many IGCs in the ocean and atmosphere propagate into a stratified fluid, traditional assessments of GC and IGC dynamics neglect energy losses to internal wave motion. We present an experimental study of the internal wave field propagating ahead of a well-mixed intrusion into a constantly stratified ambient fluid. Intrusions at various heights in the ambient fluid are examined, and synthetic schlieren imaging techniques are used to quantify wave motion ahead of the current. We note a strong influence of the level of IGC propagation on the forcing of particular supercritical wave modes, and estimate the associated energy fluxes. To construct a more balanced energy budget of intrusions into stratified environments, we compare these losses to the initial Available Potential Energy (APE) of the system and to estimates of the kinetic energy of the IGC.

[1]Supported by the National Science Foundation.

9:31AM AR.00008 Transient adjustment of UFAD systems in demand response operations , JONG KEUN YU, PAUL LINDEN, UCSD — Transient responses of a UFAD system due to Demand Response (DR) are investigated theoretically and experimentally. DR activities can be categorized by reducing thermal loads and increasing room setpoint temperature, which change the thermal environment in a room and can cause occupant thermal discomfort. By comparing the filling box time (Baines & Turner 1968) and the replenishment time in which all the air in the enclosure is replaced by supply air, non-dimensional models of UFAD systems are proposed and validated by laboratory experiments using a salt-water analogy. Two-layer stratification in the model allows us to estimate the temporal temperature change in the occupied zone and the interface height. Various DR activities, adjusting thermal loads and room setpoint temperature, are simulated to reveal the dynamic thermal responses. This study suggests that the interfacial height quickly converges to steady state compared to occupied zone temperature. The experiments show good agreement with the theoretical predictions of DR responses.

9:44AM AR.00009 ABSTRACT WITHDRAWN –

9:57AM AR.00010 The role of diffusion in natural displacement ventilation , NIGEL KAYE, Clemson University, MORRIS FLYNN, University of Alberta — The classic natural displacement ventilation model of Linden *et al.* (1990) predicts the formation of a two layer stratification when a single thermal plume is introduced into a room with vents at floor and ceiling level. The model assumes that molecular diffusion plays no role in the development of the rooms ambient stratification as such diffusion is a slow process and the plume entrainment field will act to thin the interface between the warm upper layer and cool lower layer. The prediction of a sharp interface has been confirmed by small scale salt bath experiments. However, full scale measurements and CFD simulations at larger scale indicate that the interface between the two layers is not sharp but smeared out over a finite thickness. We present two simple models for predicting the thickness of the interface as a function of the room height, floor area and vent area as well as the plume buoyancy flux and the thermal diffusivity of the fluid. The interface increases in thickness with increasing room floor area and decreasing plume strength. Our model is compared to interface thickness measurements based on CFD simulations and salt bath models and is shown to agree both phenomenologically and numerically.

Linden, Lane-Serf, & Smeed, (1990) 'Emptying filling boxes, the fluid mechanics of natural ventilation' J. Fluid Mech. **212** pp. 309–335.

Sunday, November 22, 2009 8:00AM - 10:10AM –

Session AS Geophysical: Atmospheric I 200G

8:00AM AS.00001 Effects of Release Characteristics on Urban Contaminant Dispersal , A.J. WACHTOR, University of California, Irvine, F.F. GRINSTEIN, Los Alamos National Laboratory, H.J. CATRAKIS, University of California, Irvine — The release of a chemical, biological, or radioactive contaminant in an urban environment is of particular interest due to the high population densities in urban areas. The wind flow that transports the contaminant through the urban setting is highly complex and exhibits a wide range of multi-scale phenomena. Studies of urban flows can provide information that can be of critical importance to city, state, and federal officials for creating risk management plans. Classical field experiments measuring the dispersion of scalars in urban environments provide only rather limited results. Computational experiments have the advantage of being able to offer greater insight and knowledge about the three-dimensional flow physics than field experiments are able to provide. Implicit Large Eddy Simulation (ILES) is currently a promising computational method to obtain reasonable results of urban flows. ILES resolves the large scale flow features and relies on inherent numerical dissipation to model energy transfer from the resolved scales to the sub-grid scales. Since it is the large scale dispersion of the contaminant that is of key interest, ILES is particularly well suited for this application. NRL's FAST3D-CT model based on ILES is used to simulate scalar contaminant transport in a complex urban setting. We present a study of the effects that location and associated potential temperature of the scalar contaminant release have on the subsequent dispersion of that scalar within the specified urban geometry.

8:13AM AS.00002 Temporal behavior of topographic wave-breaking , OLIVIER EIFF, Université de Toulouse, IMFT, NICOLAS BOULANGER, CNRM; CNRS-GAME, KARINE LEROUX, Université de Toulouse, IMFT, ALEXANDRE PACI, CNRM; CNRS-GAME — At low Froude numbers, the internal waves generated by flow over an obstacle or mountain will overturn and break. In the atmosphere, this results in high altitude clear air turbulence but also affects the flow field below, the most commonly known effect being the acceleration the downslope winds. Surprisingly litte is known, however, of the dynamics of the wave breaking itself. Afanasyev and Peltier (JAS, vol. 55, 1998) investigated the wave breaking region via LES and Eiff et al. (DAO, vol. 40, 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 hydraulic 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.

8:26AM AS.00003 Satellite observations of atmospheric water vapor distributions , KYLE PRESSEL, The University of California, Berkeley, WILLIAM COLLINS, Lawrence Berkeley National Laboratory — The Intergovernmental Panel on Climate Change Fourth Assessment Report identified cloud feedback as the largest source of uncertainty in Global Climate Model (GCM) estimates of climate sensitivity. Cloud feedback is resultant from the sensitivity of clouds to the thermodynamic structure of the atmosphere which is in turn modified by the clouds themselves. Prognostic statistical cloud schemes have been developed to account for subgrid-scale cloud variability in a more physically consistent manner. Statistical cloud schemes assume a distributional form for some measure of water substance concentration and then determine cloud cover and properties based on a particular parameterization of that distribution. As the majority of atmospheric water substance exists in a vapor state, we will report preliminary results of a characterization of water vapor distributions based on retrievals from the Advanced Infrared Sounder (AIRS) onboard NASA's Aqua satellite. We will report on the vertical variation of distributional forms with height and comment on the physical mechanisms maintaining these distributions.

8:39AM AS.00004 Experimental study of the effect of turbulence on the dynamics of sedimenting inertial particles[1] , COLIN BATESON, ALBERTO MOLINA, ALBERTO ALISEDA, University of Washington, HOSSEIN PARISHANI, LIAN PING WANG, University of Delaware, WOJCIECH GRABOWSKI, NCAR — Understanding the dynamics and mutual interactions of droplets in turbulent flows is important to many engineering and environmental problems including fuel injector sprays, warm rain formation, and the mass and energy transfer between the ocean and the atmosphere. Specifically, the collision and coalescence in turbulent flows is considered a key element for the growth of condensation droplets into a size range where gravitational settling mechanism can take over to produce rain drops. We study experimentally the effect of turbulence on the collision-coalescence of water droplets over a parameter range relevant to rain formation. Droplets in a size range between 1 and 40 microns are injected inside a low speed wind tunnel through an array of atomizers located at the nodes of a turbulence-inducing grid that covers the tunnel's cross section with uniform spacing. The evolution of the droplet size distribution, concentration and settling velocity is measured along the wind tunnel's test section. We will present a comparison between experimental measurements of the one and two dimensional droplet radial distribution functions and collision statistics against equivalent quantities computed from a three dimensional numerical simulation performed and presented here by Wang et al.

[1]Supported by NSF grant ATM-0731248.

8:52AM AS.00005 Statistics of Small-Scale Velocity Fluctuations and Internal Intermittency in Stratocumulus Clouds , RAYMOND SHAW, Department of Physics, Michigan Technological University, HOLGER SIEBERT, Leibniz Institute for Tropospheric Research, ZELLMAN WARHAFT, Mechanical and Aerospace Engineering, Cornell University — Clouds are known to be turbulent but the details of their internal turbulent structure have been largely unexplored. Measurements of turbulent velocities in stratocumulus clouds reveal an intermittent structure consistent with that observed in classic homogeneous isotropic turbulence. The measurements were taken using a hotwire anemometer on the helicopter-borne ACTOS measurement system. Hotwire signal artifacts resulting from droplet impacts are removed without significantly degrading the signal, such that high-order velocity structure functions can be evaluated. The structure function analysis for orders 2 through 8 show statistically significant departures from the Kolmogorov 1941 scaling, yielding scaling exponents consistent with the Kolmogorov-Obukhov refined similarity hypothesis with an intermittency exponent of 0.25. We find no evidence of any departure from the large body of knowledge obtained from the laboratory on the fine scale turbulence structure. This suggests that processes depending on the fine-scale structure of turbulence that cannot presently be measured in clouds can be explored in the laboratory setting.

9:05AM AS.00006 Remote flow sensing of complex systems: steps towards spatio-temporal prediction of flow patterns[1] , BRUNO MONNIER, PARITOSH MOKHASI, DIETMAR REMPFER, CANDACE WARK, Illinois Institute of Technology — Prediction of the spatial and temporal phenomena of wind flow patterns through urban areas is investigated. Typically sparse measurements are used in wind forecasting models for updating and prediction via a method called variational data assimilation. To improve upon this method, an experimental investigation combining various measurement tools (Hot Wire Anemometry, Stereoscopic Particle Image Velocimetry SPIV), static pressure measurements and Laser Doppler Velocimetry(LDV)) is carried out to study the airflow around wall mounted obstacles in a turbulent boundary layer. The method of Proper Orthogonal Decomposition (POD) is used to decompose the flow field into a finite set of POD coefficients which vary only in time associated with a corresponding set of POD basis functions which vary only in space. Direct measurement models utilizing the measurements from SPIV and LDV, along with indirect measurement models using sparse measurements from microphones are investigated and may ultimately be combined with state-space models to obtain more robust dynamical models.

[1]This work is supported in part by the Illinois Space Grant Consortium.

9:18AM AS.00007 Wind speed and direction measurements using the sphere anemometer , HENDRIK HEISSELMANN, MICHAEL HOELLING, JOACHIM PEINKE, University of Oldenburg & ForWind — In times of growing energy demand, the importance of wind energy is rapidly increasing and so is the need for accurate wind speed and direction measurements. The widely spread cup anemometers show significant over-speeding under turbulent wind conditions as inherent in atmospherical flows while being solely capable of detecting the wind speed. Therefore, we propose the newly developed sphere anemometer as a simple an robust sensor for direction and velocity measurements. The sphere anemometer exploits the velocity-dependent deflection of a tube, which is the order of μm and can be detected by means of a light pointer as used in atomic force microscopes. In comparative measurements under laboratory conditions the sphere anemometer showed a significantly higher temporal resolution then cup anemometers while it does not exhibit any over-speeding. Additionally, results of atmospherical wind measurements with the sphere anemometer and state-of-the-art cup anemometry are presented.

9:31AM AS.00008 Experimental study of *starting plumes* simulating cumulus cloud flows in the atmosphere , DUVVURI SUBRAHMANYAM, K.R. SREENIVAS, G.S. BHAT, S.S. DIWAN, RODDAM NARASIMHA — Turbulent jets and plumes subjected to off-source volumetric heating have been studied experimentally and numerically by Narasimha and co-workers and others over the past two decades. The off-source heating attempts to simulate the latent heat release that occurs in cumulus clouds on condensation of water vapour. This heat release plays a crucial role in determining the overall cloud shape among other things. Previous studies investigated steady state jets and plumes that had attained similarity upstream of heat injection. A better understanding and appreciation of the fluid dynamics of cumulus clouds should be possible by study of *starting plumes*. Experiments have been set up at JNCASR (Bangalore) using experimental techniques developed previously but incorporating various improvements. Till date, experiments have been performed on plumes at Re of 1000 and 2250, with three different heating levels in each case. Axial sections of the flow have been studied using standard PLIF techniques. The flow visualization provides us with data on the temporal evolution of the *starting plume*. It is observed that the broad nature of the effect of off-source heating on the *starting plumes* is generally consistent with the results obtained previously on steady state flows. More complete results and a critical discussion will be presented at the upcoming meeting.

9:44AM AS.00009 Attracting structures in volcanic ash transport , JIFENG PENG, University of Alaska Fairbanks — Volcanic eruptions and ash clouds are a natural hazard that poses direct threats to aviation safety. They may also affect human and ecosystem health. Many transport and dispersion models have been developed to forecast trajectories of volcanic ash clouds, as well as to plan safety measures. Predictions based on these models are heavily dependent on initial parameters of ash clouds, e.g., location, height, particle size and density distribution, water vs. ash content, etc. However, these initial parameters are usually difficult to determine, leading to possible inaccurate predictions of ash clouds trajectories. In this study, a dynamical systems approach is combined with volcanic ash transport models to help improve prediction. A type of attracting structures in volcanic ash transport is identified. These structures act as attractors in volcanic ash transport, and they are independent of initial parameters of specific volcanic eruptions. The attracting structures are associated with hazard zones with high concentrations of volcanic ash. And the prediction in hazard maps can be used to plan flight route diversions and ground evacuations.

9:57AM AS.00010 Mathematical Model for the Behavior of Wildfires , KEVIN DELBENE, DONALD DREW, Rensselaer Polytechnic Institute — Wildfires have been a long-standing problem in today's society. In this paper, we derive and solve a fluid dynamics model to study a specific type of wildfire, namely, a two dimensional flow around a concentrated line of fire, resulting in a narrow plume of hot gas rising and entraining the surrounding air. The model assumes that the surrounding air is constant density and irrotational, and uses an unsteady plume model to describe the evolution of the mass, momentum and energy inside the plume, with sources derived to model mixing in the style of Morton, Taylor, and Turner (Proc. Roy. Soc. London, A 234, 1-23, 1956). The sources to the dynamical processes in the plume couple to the motion through the surrounding air through a Biot-Savart integral formulation to solve the equations of motion with a line of singularities along the plume. The singularities model a vortex sheet in the same manner as Alben and Shelley (Phys. Rev. Letters, 100, 074301, 2008), except that we include a sink term in the Biot-Savart integral to couple the entrainment. The results show that this model is capable of capturing a complicated interaction of the plume with the surrounding air.

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AT Vortex Dynamics and Vortex Flows I 200H

8:00AM AT.00001 Harvesting energy in the wake of a circular cylinder using piezoelectric materials , DOGUS H. AKAYDIN, NIELL ELVIN, YIANNIS ANDREOPOULOS, City College of New York, Mechanical Engineering — The voltage generated by short, flexible piezoelectric cantilever beams placed inside turbulent wakes of circular cylinders at Reynolds numbers of 10,000 is investigated experimentally and computationally. The coherent vortical structures present in this flow generate a periodic forcing on the beam which when tuned to its resonant frequency produces maximum output voltage. There are two mechanisms which contribute to the driving forcing of the beam. The first mechanism is the impingement of induced flow by the passing vortices on one side of the beam and second is the low pressure core region of the vortices which is present at the opposite side of the beam. The sequence of these two mechanisms combined with the resonating conditions of the beam generated maximum energy output which was also found to vary with the location in the wake. The maximum power output was measured at about two diameters downstream of the cylinder. This power drops off the center line of the wake and decays with downstream distance as $(x/D)^{-3/2}$. A three-way coupled interaction simulation that takes into account the aerodynamics, structural vibration and electrical response of the piezoelectric generator has been developed.

8:13AM AT.00002 Integrability and Chaos in Body-Vortex Interactions[1] , JOHAN ROENBY, HASSAN AREF[2], Technical University of Denmark — We explore the class of dynamical systems consisting of a rigid body and N point vortices in an ideal, unbounded, 2D fluid. The body is represented by a closed curve and is free to move in response to the fluid motion. It may have a prescribed circulation about it, which is conserved. The vortices have fixed strengths and are intended to model vortices that have been shed by the body or elsewhere in the flow field. The flow at any given time and position is determined by the instantaneous vortex and body positions together with the instantaneous linear and angular velocity of the body. The equations of motion may be cast in Hamiltonian form. We analyze the equations of motion using techniques from the theory of dynamical systems. The simplest such system, a single point vortex and a circular body, is integrable. As we add vortices, or change other features of the system such as the body shape, the motion may become chaotic. Numerical solutions are shown and analyzed with an emphasis on the transition to chaos and its physical meaning. This class of systems provides a rich family of few-degree-of-freedom systems that capture essential fluid-body interaction physics.

[1]Supported by the Danish National Research Foundation through a Niels Bohr Visiting Professorship

[2]Permanent address: Virginia Tech

8:26AM AT.00003 Vortex suppression in the wake of counter rotating cylinders , PETER DEWEY, ALEXANDER J. SMITS, Princeton University — Digital particle image velocimetry is used to study the flow past a pair of counter rotating cylinders placed side-by-side normal to the freestream flow direction. The Reynolds numbers based on cylinder diameter is varied from 100 to 200 and gap-to-diameter ratios of 1, 3 and 5 are considered. An unsteady wake consisting of a pair of von Kármán vortex streets is present in the flow field when the cylinders are rotated below a critical value. Above this critical value, counter rotation of the cylinders suppresses vortex formation. The critical rotational speed varies only slightly with Reynolds number but exhibits a strong dependence on the gap-to-diameter ratio. As the gap-to-diameter ratio increases, the critical rotational speed approaches values expected to suppress vortex formation for a single rotating cylinder, indicating that the wakes of the cylinder pair have more interaction for small gap-to-diameter ratios. At sufficiently high rotational speeds the streamlines around the cylinder pair resemble a doublet potential flow. The experiments were inspired by the computations performed by Andy Chan and Antony Jameson at Stanford University.

8:39AM AT.00004 Flow Development over a Circular Cylinder with a Stepwise Discontinuity , CHRIS MORTON, SERHIY YARUSEVYCH, University of Waterloo — Cross-flow around a step cylinder is common in various engineering applications, for example, heat exchangers and buildings, where understanding flow development is often of critical importance for engineering design. Moreover, the relatively simple geometry of a step cylinder allows modeling complex vortex interactions. For a step cylinder in uniform flow, the flow development is dependent on the Reynolds number (Re_D) and the ratio of the large cylinder diameter (D) to the small cylinder diameter (d). In this study, vortex shedding phenomena occurring in the wake of a step cylinder is investigated using an unsteady RANS based numerical approach for $Re_D = 300$ and D/d = 2. Based on the numerical results, three distinct spanwise vortex cells were identified in the step cylinder wake: one vortex shedding cell in the wake of the small cylinder and two vortex shedding cells in the wake of the large cylinder. A comparative analysis with available experimental data showed that the numerical simulations adequately modeled wake vortex development and interactions in the near wake region. One of the vortex cells forming downstream of the step was found to have a cyclic appearance, with the periodicity being linked to downwash fluctuations near the step. In addition, the results suggest that streamwise vortices develop at the step and unsteady interactions between the streamwise and spanwise vortices occur in the near wake.

8:52AM AT.00005 Three dimensional flow around a flexible circular cylinder in cross-flow , FRANCISCO HUERA-HUARTE, DAVID JEON, MORTEZA GHARIB, California Institute of Technology — The three dimensional flow around a flexible cantilever model undergoing vortex-induced vibrations has been studied by using Defocusing Digital Particle Image Velocimetry (DDPIV). The DDPIV technique allowed the simultaneous measurement of the motion and the flow around a portion of the cylinder. Different circular cylinder models accounted for several high aspect (length over diameter) and low mass ratios (mass over mass of displaced fluid), leading to flow-induced vibrations with different dominant mode shapes and frequencies. The quantitative study of the wake structures of cylinders able to vibrate at different structural modes is of particular interest as there is a lack of published results. Moreover, the mechanisms yielding to mode locked-on behavior and the topology of the wake under this situation are poorly understood.

9:05AM AT.00006 Vortex-Body Interaction using a Level-Set Based Overset Grid Method[1] , SIMTHA RENJITHAM, JEFFREY MARSHALL, University of Vermont — An overset grid method is presented for solution of the integral vorticity-velocity formulation of the Navier-Stokes equations. The method uses an inner body-fitted grid and an outer Cartesian grid. The Biot-Savart integral is solved using an adaptive, optimized multipole acceleration method. The integration is performed over all inner grid cells, over all "active cells" of the outer grid that lie entirely outside of the inner grid, and over sub-elements of a set of overhanging cells of the outer grid that overlap part of the inner grid. A level-set function is introduced in which the zero level-set curve coincides with the outer surface of the inner grid. This level-set function is used to rapidly subdivide the overhanging grid cells into triangular sub-cells which lie entirely outside of the inner grid, while omitting the parts of these cells that lie inside the inner grid, so as to avoid double-counting the vorticity in these regions. The pressure is solved as a post-processing variable using a boundary-element formation that requires evaluation of an integral using a parallel method to that used for velocity calculation. The method is applied to two-dimensional flow past stationary and moving bodies, and it is well suited to vortex-body interaction with complex, moving bodies.

[1]This project was supported by the U.S. Army Research Office.

9:18AM AT.00007 On the frequency of high Reynolds number shedding in bluff-body wakes , FERNANDO PONTA, Michigan Technological University — In this talk we shall explore the physical phenomenon of vortex shedding at high and extra-high Reynolds number. We start from a previous work where Ponta and Aref introduced a rationale for the empirically observed Strouhal-Reynolds number relationship for vortex shedding at low Reynolds. Analyzing the turbulent transport of momentum, the rationale is extended to high Reynolds number regimes. Results compared satisfactorily with the existent experimental evidence, and their extension to extra-high- Reynolds geophysical flows will be discussed.

9:31AM AT.00008 Using a hybrid cyber-physical system in the study of body motion due to vortex dynamics , A. MACKOWSKI, C.H.K. WILLIAMSON, Cornell University — We are interested in the effect of vortex dynamics causing vibration of bodies in a flow. In these studies, one needs to select essential parameters for the body, such as mass, spring stiffness, and damping. Normally, these parameters are set physically by selecting mechanical elements. However, in our approach, which utilizes a computer-controlled XYΘ towing tank and a force-feedback control system, we impose mass- spring-damping parameters in virtual space and in three degrees of freedom. [A similar concept, in one degree of freedom, was pioneered by a group at MIT (Miller 1996; Hover, Techet, Triantafyllou 1997), in studies of vortex-induced vibration of cables.] Although the use of a cyber-physical system has clear advantages over using a fixed, physical experiment, there are serious challenges to overcome in the design of the governing control system. The presence of noise in the dynamic force measurements and the effects of a finite time delay in controller response cause problems both for the implementation and physical accuracy of such a setup. In this presentation, we explore a new methodology for creating a controller suitable for systems with several degrees of freedom. Our controller, based on a discretization of Newton's laws, makes it straightforward to add and modify any kind of nonlinear, time-varying, or directional force, virtually. We shall present applications of this approach to problems in flow-induced vibration.

9:44AM AT.00009 Vortex-induced vibrations of an elastically mounted sphere at Re = 300: Hysteresis and vortex shedding modes[1] , SURESH BEHARA, IMAN BORAZJANI, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, University of Minnesota — We carry out fluid-structure interaction (FSI) simulations to investigate the excitation mechanisms and vortex shedding modes of an elastically mounted sphere that is free to oscillate in all three directions using the FSI-CURVIB method [Borazjani et al, J. Comp. Physics, 2008]. The simulations are performed for Re=300 over a range of reduced velocities. We report novel results showing hysteresis in the response curve depending on whether the reduced velocity is decreased or increased. Large amplitude oscillations are found to persist even for small reduced velocities when the reduced velocity is decreased from higher values. Increasing the reduced velocity from low values, on the other hand, causes the large-amplitude oscillations to be excited only at higher reduced velocities. Our simulations elucidate the 3D wake structures associated with each hysteresis branch and reveal a new vortex shedding mode. We show that the lower hysteresis branch exhibits the standard braided-hairpin wake mode while the upper branch exhibits a striking wake structure characterized by intertwined, longitudinal spiral vortices.

[1]This work was supported by NSF Grant 0625976, NIH R01-HL-07262 and the Minnesota Supercomputing Institute.

9:57AM AT.00010 Vortex shedding interactions with an oscillating flat plate[1] , ARNOLD SONG, KENNETH BREUER, Brown University — We present results from a model system designed to study the interactions between vortex shedding and structural compliance, as might be exhibited in systems as diverse as flying animals with compliant wings or traffic signs subject to hurricane-force winds. A sharp-edged plate is mounted at high angle of attack such that vortex shedding from the leading and trailing edges results in fluctuations of the aerodynamic forces. In its open-loop mode of operation, the angle of the plate is oscillated in a controlled sinusoidal manner, and the aerodynamic forces and vortex characteristics are measured using a torque sensor at the root of the support rod and a hot wire located in the wake. The onset of hysteresis in the aerodynamic forces generated during the pitching cycle is documented as a function of mean and fluctuating angles. In its closed-loop mode, the angle of the plate becomes a function of the aerodynamic forces such that an arbitrary virtual stiffness and damping can be proscribed. These different modes of operation, generated by the interactions between the fluid and structural forces are presented and discussed.

[1]Supported by AFOSR - bioinspired flight program

Sunday, November 22, 2009 8:00AM - 9:57AM –
Session AU Granular I: Jamming I 200I

8:00AM AU.00001 Jamming of Granular Materials in Wedge Hoppers[1] , SUMMER SARAF, SCOTT FRANKLIN, Rochester Institute of Technology — We study the jamming of ordinary and rod-like granular materials in wedge-shaped hoppers and compare the probability distributions for exit mass with those obtained from cylindrical hoppers. While cylindrical hoppers show an exponential probability distribution, we find that the rectangular exit aperture of wedge-shaped hoppers exhibit a power law decay for both spheres and rods. This behavior can be explained with a model of the rectangular exit aperture as composed of a series of round, adjacent apertures each with a statistically independent jamming probability. We speculate that the spatially varying jamming probability results from inhomogeneities in the granular material, with regions of the material more tightly packed than others, and thus more likely to jam.

[1]This work is funded in part by the National Science Foundation, DMR#0706353, and Research Corporation #CC5960.

8:13AM AU.00002 Jamming of Granular Flow in a Two-Dimensional Hopper[1] , JUNYAO TANG[2], SEPEHR SADIGHPOUR[3], ROBERT BEHRINGER[4], Duke Physics — We seek an understanding of the physics of jamming in flow from a hopper. Using spatio-temporal video data for photoelastic disks (mean diameter d) flowing through a two-dimensional hopper (opening size D.), we have found experimental support for the hypothesis that the probability of flow surviving until time t without jamming has the form $P_s(t) = \exp(-t/\tau)$. The important physics is encapsulated in τ, and how that depends on the ratio D/d. Estimates of τ vary as a power-law or an exponential in D/d for a jamming model and an arch formation model. Through particle tracking we conclude that jamming requires both a high packing fraction and a stable force chain arch at the outlet. Work in progress is yielding data for τ vs. the hopper angle as well as D/d.

[1]This work has been supported by IFPRI.
[2]Graduate Student
[3]Graduate Student
[4]Advisor

8:26AM AU.00003 Two-dimensional dense granular material subject to uniform simple shear[1] , JIE REN, R. BEHRINGER, Duke University — We have performed 2D shear experiments using photoelastic particles and a novel apparatus to investigate the role of shear strain on the jamming of a dense granular material. The goal of this work is to explore the shear stress axis in the jamming phase diagram proposed by Liu and Nagel. The experiments are carried out using a 2D simple shearing apparatus, whose base is specifically designed to perform nearly uniform shear at the particle scale across the shearing area. By using photoelastic particles, we can measure stresses, strains, contact forces, and particle displacements while shearing. We note two important observations from this work: 1) for densities that are below the value for isotropic jamming, the system jams when subject to shear strain; 2) starting from jammed isotropic states at higher densities, the application of quasi-static shear at constant density does not lead to failure, but rather a strengthening of the system.

[1]Work of RPB supported by NSF grant DMR0906908, ARO grant W911NF-07-1-0131-00, and BSF grant 2004391.

8:39AM AU.00004 Topology of force chains in dense granular materials[1] , LOU KONDIC, YIGUANG YAN, NJIT, MIROSLAV KRAMAR, KONSTANTIN MISCHAIKOW, Rutgers — Force chain structures are well known and well researched due to their importance in determining static and dynamic features of dense particulate systems. However, so far there is no well defined approach towards understanding properties of these structures and distinguishing them in different systems. In this talk, we will present novel approach 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. While the present talk will concentrate on the results of discrete element simulations, we will see that this new approach has a significant potential in comparing experimental and theoretical results in a well defined and precise manner.

[1]Supported by NSF grant DMS-0835611.

8:52AM AU.00005 Friction-induced hysteresis in quasi-static granular jamming , MAHESH BANDI, Los Alamos National Laboratory, ANDRAS LIBAL, Universiteit Antwerpen, MICHAEL RIVERA, ROBERT ECKE, Los Alamos National Laboratory — Static granular packings are usually interrogated via quasi-static measurements where the packing fraction serves as the control parameter to study the pack evolution. In the absence of externally induced vibrations (effective granular temperature), quasi-static measurements are justified because the system in question is athermal. Whereas this is true for frictionless granular packings, we experimentally demonstrate the failure of quasi-staticity for frictional packings in a quasi two-dimensional system of disks. This failure is traced to hysteretic responses in the system which shifts the critical packing fraction at which the system jams to progressively higher values as the system is repeatedly jammed and un-jammed. The shift in critical packing fraction marks the system's evolution from a Random Loose Packed (RLP) to a Random Close Packed (RCP) density. This rate of evolution is experimentally determined to depend upon the quasi-static step size and the static friction coefficient of the constituent disks in the system.

9:05AM AU.00006 Is random close packing of beads well defined? , FRANK RIETZ, University Magdeburg, CHARLES RADIN, HARRY L. SWINNEY, UT Austin, MATTHIAS SCHROETER, Max-Planck-Institute Goettingen — The name random close packing refers to the experimental observation that some ways of packing of monodisperse beads (like vertical vibration or sedimentation) can't exceed a volume fraction of ≈64%. There are several competing theories for this phenomenon [1-3]. However, it is possible to surpass the random close packing limit by cyclic shearing [4]. We investigate the three-dimensional distribution of particles in such a shear cell. Index matching of the surrounding liquid provides access to the interior of the granular bed. A laser sheet is scanned through the sample and by adding a fluorescent dye to the liquid we can determine the particle positions. The experiment starts at packing fractions well below random close packing. After a few thousand cycles packing fractions above 64% are achieved. By determination of Voronoi cells we characterize the local packing densities and measure order parameters around the onset of random close packing. This allows us to comment on the question if random close packing is well defined. [1] Torquato; Phys. Rev. Lett. 84, 2064 (2000). [2] Kamien; Phys. Rev. Lett. 99, 155501 (2007). [3] Radin; J. Stat. Phys. 131, 567 (2008). [4] Nicolas; Eur. Phys. J. E 3, 309 (2000).

9:18AM AU.00007 Random Packings of Rod-like Granular Materials[1] , SCOTT FRANKLIN, Rochester Institute of Technology — Piles of large aspect ratio granular materials are known to form solid plugs, significantly more rigid than piles of ordinary sand or rice. We create random, jammed packings of spherocylinders — cylinders with hemispherical endcaps — using energy minimization techniques. The packing fraction at high aspect ratios agrees with a mean-field model that scales as the inverse of excluded volume, implying that that contact number is constant even for very long, thin particles. This is confirmed by a direct analysis of the average contact number at large aspect ratios. The structure of the jammed state can be investigated through the dynamical matrix of elastic modes. In contrast with other work on ellipsoids, our packings show low energy translational modes and higher energy rotational modes, which we explain as resulting due to the absence of particle curvature.

[1]This work funded by the National Science Foundation DMR #0706353.

9:31AM AU.00008 Jamming of granular materials under shear[1] , JIE ZHANG, ROBERT BEHRINGER, Duke University — We probe the transition of a disordered system between an unstable (e.g. fluid-like) state and a stable (e.g. solid-like or jammed) state. Examples of relevant systems include glasses, foams, colloids and granular materials. Liu and Nagle proposed a jamming diagram with axes of inverse density, temperature and shear stress, and a region near the origin was proposed to encompass the jammed states. Point J on the diagram, (isotropic jamming) was thought to be the lowest possible jammed density, and for denser systems, shear stress was thought to lead to unjamming. Recent work has focusd on isotropic jamming. Here, we explore the effect of shear on jamming. We have carried out experiments using quasi-2D systems of photoelastic disks subject to pure shear. We obtain inter-particle contact forces as well as other key information. From this data we compute stresses, densities, etc. Contrary to the above picture, we find that the application of shear to densities lower than that at point J can lead to jammed states. Shear applied to isotropic jammed states does not lead to unjamming, but rather to an increase in all stresses. These data, which obviously pertain to frictional particles, suggest a jamming diagram given by shear stress, pressure and inverse density.

[1]Work supported by NSF grant DMR0906908, ARO grant W911NF-07-1-0131-00, and BSF grant 2004391.

9:44AM AU.00009 A kinetic theory of plastic flows in jammed materials[1] , LYDERIC BOCQUET[2], University of Lyon, France, ANNIE COLIN, LOF, University Bordeaux 1, ARMAND AJDARI, ESPCI, Paris, France — Amorphous jammed materials of diverse nature display complex flow properties intermediate between solid and liquid, as characterized by the existence of a yield stress. Furthermore flows in such systems usually exhibit spatial inhomogeneities, which cannot be reconciliated with classical rheological descriptions. We present a novel kinetic approach for the elasto-plastic flow dynamics of jammed materials, describing the spatio-temporal collective dynamics of the localized plastic events occuring during the flow [1]. This description yields a non-local constitutive law for the flow, introducing as a key dynamic quantity the local rate of plastic events. This quantity, interpreted as a local fluidity, is spatially correlated with a correlation length diverging in the quasistatic limit, i.e., close to yielding. In line with recent experimental [2] and numerical observations, we predict finite size effects in the flow behavior, as well as the absence of an intrinsic local flow curve.

[1] L. Bocquet, A. Colin, A. Ajdari, *Phys. Rev. Lett.* (2009) in press
[2] J. Goyon, A. Colin, G. Ovarlez, A. Ajdari, L. Bocquet, *Nature* **454** 84 (2008)

[1]This work is supported by ANR program SYSCOMM.
[2]http://lpmcn.univ-lyon1.fr/∼lbocquet

Sunday, November 22, 2009 8:00AM - 10:10AM –
Session AV Flight I 205A-D

8:00AM AV.00001 Optimization Study for Hovering Flapping Flight[1] , HUMBERTO BOCANEGRA EVANS, JAMES J. ALLEN, New Mexico State University, B.J. BALAKUMAR, Los Alamos National Laboratory — A scaled robotic hummingbird model was used to perform a flow analysis of hovering flight at a range of Reynolds numbers (1,750$< Re <$25,000). The flow structure was analyzed using Particle Image Velocimetry and force data was acquired through a strain gage load cell. An attached leading-edge vortex (LEV) consistently appears at the mid-downstroke on the PIV data, which differs from the notion of LEV shedding at higher Reynolds numbers. The coefficient of lift was calculated through the circulation of the flow and compared to the values yielded by the load cell. A peak in the coefficient of lift calculated from PIV data through circulation is in agreement with the Reynolds number at which a rufous hummingbird hovers ($Re \approx 3600$), which suggests that hummingbirds hover in a highly efficient manner.

[1]Funded by ARO and STTR.

8:13AM AV.00002 Flapping counter torque (FCT) in animal flight: Experimental results and mathematical models , BO CHENG, XINYAN DENG, University of Delaware — From our previous studies on a range of insects from fruit flies to cockatoos during fast yaw turning maneuvers (body saccades), we found that body rotation causes a substantial aerodynamic counter torque, termed as flapping counter-torque (FCT), which acts in the opposite direction of turning. In this study, we show that FCT exists in all roll, pitch and yaw axes and are linearly dependent on the flapping frequency and rotational velocity, respectively. We measured the FCTs systematically (by varying wing beat frequency and body turning velocity) on a pair of dynamically scaled robotic model wings. Furthermore, we developed mathematical FCT models based on quasi-steady analysis for roll, pitch and yaw axes. The results show that the experimental data matches the prediction of the analytical models. FCT induced passive damping accounts for a large part of the deceleration in saccade of animal flight, and implies passive rotational stability of the angular velocity dynamics in flapping flight.

8:26AM AV.00003 Controlling Pitching Instability in 3D Flapping Flight , SONG CHANG, Applied and Engineering Physics, Cornell University, Ithaca, NY 14853, JANE WANG, Mechanical and Aerospace Engineering, Cornell University, Ithaca, NY 14853 — Flying insects actively control their wings to maintain the stability in steady flight as well as to execute maneuvers. The control strategies depend on the coupling of sensory feedback loops of insects and the underlying dynamics of the 3D flapping flight. In this talk, we first present a general method for efficiently simulating the 3D flapping flight of the coupled wing-body system in the quasi-steady limit. We then quantify the stability of the periodic solutions that correspond to equilibrium flight. The analysis shows that the flapping system exhibits an inherent instability in pitching, and this instability can be further understood in a reduced-order model. We propose a simple control strategy for stabilizing the pitching by modulating wing motions.

8:39AM AV.00004 Fruit flies use flight auto-stabilization to recover from aerial "stumbles" , LEIF RISTROPH, ATTILA BERGOU, Cornell University, GUNNAR RISTROPH, Ascendant Engineering, KATHERINE COUMES, GORDON BERMAN, JOHN GUCKENHEIMER, Z. JANE WANG, ITAI COHEN, Cornell University — Just as manned flight was made possible by control mechanisms, the flapping-wing flight of animals also relies on strategies that ensure recovery from disturbances. Previous studies performed on tethered and dissected insects indicate that the sensory, neurological, and musculoskeletal systems play important roles in flight control. Such studies, however, have yet to produce an integrative model of flight stability since they do not incorporate the interaction of these systems with free-flight aerodynamics. Here, we directly investigate control and stability through the application of brief torques to free-flying fruit flies and measurement of their behavioral response. High-speed video and a new motion tracking method capture the aerial "stumble", and we discover that flies respond to gentle disturbances by accurately returning to their original orientation. This accurate and fast recovery motivates a feedback control model that includes the insect's ability to sense body rotations, process this information, and actuate the wing motions that generate corrective aerodynamic torque. Thus, as with modern fighter jets, the common fruit fly employs an auto-stabilization scheme that maintains its flight course and allows for navigation through complex aerial environments.

8:52AM AV.00005 Fruit flies modulate passive wing pitching to induce in-flight turns , ATTILA BERGOU, LEIF RISTROPH, JOHN GUCKENHEIMER, ITAI COHEN, JANE WANG, Cornell University — To control their ?ight, insects must have mechanisms to modulate their wing kinematics. Exactly how insects control their wing motions to execute observed flight maneuvers is poorly understood. Here, we measure the wing and body kinematics of freely flying fruit flies performing turns and, in conjunction with numerical simulations and mathematical models, probe how they control their wing motion to ultimately alter their flight path. We find that these flies induce sharp turns by applying an overall bias to the passive pitching motion of their wings. We present a simple mechanical model for the wing actuation that quantitatively predicts the turning dynamics of the insect.

9:05AM AV.00006 Flapping counter force - a unique flight stabilizing mechanism enabled by flapping wings , HU DAI, HAOXIANG LUO, Vanderbilt University, XINYAN DENG, Purdue University — The flyers in nature are more sensitive to disturbances than the much-larger airplanes, and meanwhile, many of them (e.g., insects) lack the geometrical features that airplanes typically have, e.g., the vertical/horizontal tails. Therefore, a passive flight stabilizing mechanism would be of particular importance to the biological flyers, who otherwise would have to spend a great deal of effort to actively control their flight. It was recently found that insects and other flying animals possess a unique passive stabilization mechanism that stems from the coupling between their body movement and the flapping-wing motion (Hedrick, Cheng and Deng, Science, 2009). More specifically, the unsteady movement of the flyer's body in a disturbed flight modifies the effective kinematics of the wing, creating a resistant force that counteracts the body motion. In this work, we use direct numerical simulations to compute the flapping counter force associated with a two-dimensional wing, and the transient process of the disturbed body motion is also computed via flow-structure interaction. The flyer's body is represented by a lumped mass, and the flow around the wing is resolved by the simulations to accurately account for the force production mechanism. The computed force and the body transition will be compared with a quasi-steady analysis.

9:18AM AV.00007 Aerodynamics of Dragonfly in Hover: Force measurements and PIV results , XINYAN DENG, ZHENG HU, University of Delaware — We useda pair of dynamically scaled robotic dragonfly model wings to investigate the aerodynamic effects of wing-wing interaction in dragonflies. We follow the wing kinematics of real dragonflies in hover, while systematically varied the phase difference between the forewing and hindwing. Instantaneous aerodynamic forces and torques were measured on both wings, while flow visualization and PIV results were obtained. The results show that, in hovering flight, wing-wing interaction causes force reduction for both wings at most of the phase angle differences except around 0 degree (when the wings are beating in-phase).

9:31AM AV.00008 Aerodynamics of Dragonfly in Forward Flight: Force measurements and PIV results , ZHENG HU, XINYAN DENG, University of Delaware — We used a pair of dynamically scaled robotic dragonfly model wings to investigate the aerodynamic effects of wing-wing interaction in dragonflies. We follow the wing kinematics of real dragonflies in forward flight, while systematically varied the phase difference between the forewing and hindwing. Instantaneous aerodynamic forces and torques were measured on both wings, while flow visualization and PIV results were obtained. The results show that, in forward flight, wing-wing interaction always enhances the aerodynamic forces on the forewing through an upwash brought by the hindwing, while reduces the forces on the hindwing through a downwash brought by the forewing.

9:44AM AV.00009 Lift production of a hovering pyramid in an oscillatory airflow , ANNIE WEATHERS, Dept. of Physics, New York University, BRENDAN FOLIE, Dept. of Math, Harvey Mudd College, BIN LIU, STEPHEN CHILDRESS, Courant Institute, New York University, JUN ZHANG, Dept. of Physics and Courant Institute, New York University — We investigate the dynamics of rigid, hollow "pyramids" placed within a background airflow, oscillating with zero mean. The asymmetry of the body introduces a net upward force. We find that when the amplitude of the airflow is above a threshold, the net lift exceeds the weight and the object hovers. Our results show that the objects hover at far smaller air amplitudes than would be required by a quasi-steady theory. We find that paired vortices are generated during each period of the oscillatory flow, which provide the lift. We also observe that lighter objects do not necessarily hover more easily, because they tend to be entrained by the flow, reducing the relative motion and the resultant lift. In fact a finite flow amplitude is observed to be required for hovering in the limit of zero body mass.

9:57AM AV.00010 Aerodynamic force variation in an inclined hovering motion by kinematic and geometric controls[1] , HYUNGMIN PARK, HAECHEON CHOI, Seoul National University — Due to the excellent flight capability with a high maneuverability, dragonfly flight has been a great interest in various fields. In the present study, we construct a one-paired dynamically scaled dragonfly wing model, perform an inclined hovering motion by wing flapping in a white-oil tank, and measure the normal and tangential forces on the wing. First, we investigate the effect of kinematic parameters of wing motion such as the attack angle (α), pitching duration, pitching timing, etc. The Reynolds number is 1,900 or 2,430 depending on the wing shape. We find that the aerodynamic forces vary greatly with these kinematic parameters. On the other hand, the corrugation on the wing surface has been found to increase the lift force in gliding flight. In this study, we investigate the effect of surface corrugation on the force of the flapping wing. With the corrugation, the drag force slightly increases during a downstroke (high α) and the lift force increases during an upstroke (small α), respectively, resulting in the increase of the mean vertical force by $10 \sim 30\%$ depending on the wing trajectory. We further investigate the force variation by kinematic and geometric controls using flow visualization and the result will be shown in the presentation.

[1]Supported by the National Research Laboratory Program and the Korea Research Foundation Grant, MEST.

Sunday, November 22, 2009 8:00AM - 10:10AM –

Session AW Mini-Symposium on Intracellular Fluid Dynamics 208A-D

8:00AM AW.00001 Cellular hydraulics: physics and physiology , L. MAHADEVAN, Harvard University — Hydraulics (from the Greek word $\delta\rho\alpha\upsilon\lambda\iota\kappa\varsigma$) is the study of fluid movements and fluid power. In this brief talk, I will discuss the implications of water movements through soft porous structures for the dynamics of nuclear swelling and the cytoskeleton in animal cells. If time permits, I will conclude with a quantitative description of the growth of pollen tubes treated as hydraulic machines.

8:26AM AW.00002 Active Gels and Cell Quakes: Exploring the non-equilibrium rheology and fluctuation spectrum of motor-driven polymer networks , ALEX LEVINE[1], University of California, Los Angeles — Recent experiments on molecular motor driven in vitro F-Actin networks have found anomalously large strain ?uctuations at low frequency. In addition, the shear modulus of these active networks becomes as much as one hundred times larger than that of the same system in equilibrium. In this talk we develop a theory of both these phenomena using a two-?uid model of a low-density isotropic semi?exible network driven by molecular motors. Relying on only simple assumptions regarding the motor activity in the system, we find that we can quantitatively understand both the low-frequency ?uctuation enhancement and the nonequilibrium stiffening of the network. We also show the results of new numerical studies of semiflexible networks driven by molecular motors that explore the effects of high motor density in isotropic networks and the effect of nematic order in the active filament network. These results have implications for the interpretation of microrheology in such active networks including the cytoskeleton of living cells. In addition, they may form the basis for theoretical studies of biomimetic nonequilibrium gels whose mechanical properties are tunable through the control of their nonequilibrium steady-state.

[1]In collaboration with: F.C. MacKintosh (Department of Physics and Astronomy, Vrije Universiteit)

8:52AM AW.00003 Resolving the Role of Actoymyosin Contractility in Cell Microrheology , DENIS WIRTZ[1], Johns Hopkins University — Einstein's original description of Brownian motion established a direct relationship between thermally-excited random forces and the transport properties of a submicron particle in a viscous liquid. Recent work based on reconstituted actin filament networks suggests that nonthermal forces driven by the motor protein myosin II can induce large non-equilibrium fluctuations that dominate the motion of particles in cytoskeletal networks. Here, using high-resolution particle tracking, we find that thermal forces, not myosin-induced fluctuating forces, drive the motion of submicron particles embedded in the cytoskeleton of living cells. These results resolve the roles of myosin II and contractile actomyosin structures in the motion of nanoparticles lodged in the cytoplasm, reveal the biphasic mechanical architecture of adherent cells-stiff contractile stress fibers interdigitating in a network at the cell cortex and a soft actin meshwork in the body of the cell, validate the method of particle tracking-microrheology, and reconcile seemingly disparate atomic force microscopy (AFM) and particle-tracking microrheology measurements of living cells.

[1]In collaboration with: Christopher M. Hale; Sean X. Sun, Johns Hopkins University

9:18AM AW.00004 Anisotropic viscoelastic properties and cytoskeletal structure of endothelial cells subject to shear flow , JUAN C. DEL ALAMO[1], University of California, San Diego — Adherent cells remodel in response to mechanical stimuli leading to a redistribution of intracellular forces that depends on the viscoelastic properties of the cytoskeleton. We have analyzed the magnitude and anisotropy of these properties in confluent vascular endothelial cells subject to continuous flow. For this purpose we used Directional Particle Tracking Microrheology, which measures the second-order tensor of intracellular marker displacements, allowing us to determine the principal directions of highest and lowest shear modulus at each position. We studied the orientation of these principal directions relative to those of the actin stress fibers. After the application of flow shear the cells' stress fibers gradually orient parallel to the flow and the principal directions of the shear modulus become parallel and perpendicular to the flow. The role of ATP-driven myosin-II contractions in the observed anisotropy is analyzed by using cells treated with drugs inhibiting myosin-II function.

[1]In collaboration with: K. Osterday, G. Norwich, Y.-S. Li, S. Chien and J. C. Lasheras, University of California San Diego

9:44AM AW.00005 Structural response and remodeling of red blood cells - a multiscale modeling approach , QIANG ZHU[1], University of California, San Diego — A red blood cell contains cytosol enclosed inside a composite membrane consisting of a fluidic lipid bilayer reinforced by a single layer of protein skeleton. It has been demonstrated that mechanical loads can trigger dissociation of inter-protein and protein-to-lipid linkages and cause structural remodeling and failure. To understand these effects, it is vital to quantitatively characterize the mechanical forces acting within the membrane. For this purpose we developed a multiscale model to study distributions of internal stress in response to external load. In this method, the cell is modeled at three length scales: in the complete-cell level it is depicted as two layers of continuum shells, one representing the lipid bilayer and the other the skeleton; a molecular-detailed model of the skeleton is developed to predict its constitutive properties; a nonlinear stain-stretch model of Sp (a major protein in the skeleton) is applied to study the mechanical properties of the cell in large deformations. With this model we investigated mechanical responses of the system under canonical experiments such as micropipette aspirations and optical tweezer stretching. Model validations were conducted through comparisons with benchmark experiments.

[1]In collaboration with: Z. Peng and R. J. Asaro, University of California San Diego

Sunday, November 22, 2009 10:30AM - 12:27PM –
Session BA Turbulent Boundary Layers: DNS/LES 101A

10:30AM BA.00001 High-Reynolds number turbulent boundary layers studied by numerical simulation , PHILIPP SCHLATTER, QIANG LI, GEERT BRETHOUWER, ARNE V. JOHANSSON, DAN S. HENNINGSON, KTH Mechanics, Stockholm, Sweden — Direct and large-eddy simulations (DNS and LES) of spatially developing high-Reynolds number turbulent boundary layers (Re_θ up to 4300) under zero pressure gradient are studied. The inflow of the computational domain and the tripping of the boundary layer is located at low Reynolds numbers $Re_\theta \approx 350$, a position where natural transition to turbulence can be expected. The simulation thus includes the spatial evolution of the boundary layer for an extended region, providing statistics and budget terms at each streamwise position. The data is obtained with up to $O(10^{10})$ grid points using a parallelised, fully spectral method. The DNS and LES results are critically evaluated and validated, in comparison with other relevant data, *e.g.* the experiments by Österlund *et al.* (1999). Quantities difficult or even impossible to measure, *e.g.* pressure fluctuations and complete Reynolds stress budgets, shall be discussed. In addition, special emphasis is put on a further quantification of the large-scale structures appearing in the flow, and their relation to other wall-bounded flow as *e.g.* channel flow. The results clearly show that with today's computer power Reynolds numbers relevant for industrial applications can be within reach for DNS/LES.

10:43AM BA.00002 Direct numerical simulation of flat-plate turbulent boundary layer up to $Re_\theta = 2000$, XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, CTR, Stanford University — Recent flow visualization and statistical results from DNS of flat plate turbulent boundary layer are presented. We have extended the DNS of Wu & Moin (JFM, **630**, 2009) to twice the streamwise length with much increased Reynolds number. The calculations were carried out using a grid of 8192×500×256 along the streamwise, wall-normal and spanwise directions, respectively. The computational domain in the spanwise direction was increased by 50% over the earlier calculations of Wu & Moin. Well-controlled boundary layer bypass transition to turbulence is promoted by the passage of isotropic turbulent flow patches in the free-stream. As in Wu & Moin, in the transitional region, the instantaneous flow fields are vividly populated by hairpin vortices with distinct regularity. Many of these transitional hairpin vortices reach the local boundary layer edge. Throughout the turbulent region, forests of hairpin vortices are found to persist. As Reynolds number increases with downstream distance, the hairpin vortices exhibit more chaotic characteristics in the form of twisting and merging with more pronounced irregularity compared to their transitional counterparts. At higher Reynolds numbers hairpin forests are generally limited to about 70 percent of the local boundary layer thickness. Mean and second-order turbulence statistics and the corresponding spatial and time scales will also be presented.

10:56AM BA.00003 Effects of Computation Pipe Length on Turbulence Statistics using DNS of Turbulent Pipe Flow , CHENG CHIN, ANDREW OOI, IVAN MARUSIC, The University of Melbourne, HUGH BLACKBURN, Monash University — Direct numerical simulation (DNS) of fully developed turbulent pipe flow is carried out at $Re_\tau \approx 170$ and 500 (based on friction velocity, u_τ, and pipe radii, δ) to investigate the effects of computational pipe length on the turbulence statistics. Here the DNS uses a spectral scheme in the streamwise and azimuthal directions. Various turbulence statistics are compared for different pipe lengths including the mean flow, Reynolds stresses, correlations, one-dimensional energy spectra, and skewness and flatness.The results show that in the near wall region (below the buffer region, $z^+ \leq 30$ say), a required pipe length of at least O(3000) viscous wall units is required for all turbulent statistics to converge and be independent of the length. In the outer region, comparison of spectra suggest that pipe length of $6\pi\delta$ is sufficient for the results to be pipe-length independent. Preliminary results for higher order statistics suggest that longer lengths may be required.

11:09AM BA.00004 Fluctuating Vorticity in Turbulent Wall Layers , RONALD PANTON, University of Texas — DNS data for the correlations of fluctuating vorticity in the streamwise $< \omega_x\omega_x >$, spanwise $< \omega_z\omega_z >$, and normal $< \omega_y\omega_y >$ directions is given in the various papers of Del Alamo, Jimenez, Zandonade, Moser, and Hoyas (PofF **15**, L-41; JFM, **500**,p135, PofF, **18**, 011702 at four Reynolds numbers. Previously, APS Bulletin **53,** 18, 2008 EA.00004, the inner wall region was considered. It was shown that the normal component profiles at different Reynolds numbers collapse together when scaled as $< \omega_y\omega_y > / (u_*^4/\nu^2)$. However, the other components, $< \omega_x\omega_x >$ and $< \omega_z\omega_z >$, require a two-term expansion of the form $F \sim F_0 + F_1 \cdot u_*/U$. The first term scaling as $< \omega\omega >_0 / ((u_*/\nu)^2 u_* U)$ and the second scaling as $< \omega\omega >_1 / (u_*^4/\nu^2)$. In the outer region a completely different scaling is required. An analysis of the matching behavior between the two regions shows that the common part is a function that decreases as $1/y$. This implies that in the outer region the proper scaling is $< \omega\omega > / [u_*^3/(h\nu)]$. Indeed, profiles of all three components collapse in the outer region in this variable. Furthermore, all three components show a marked tendency toward the same level and isotropic behavior.

11:22AM BA.00005 DNS of Turbulent Boundary Layers with/without External Pressure Gradient Based on a Multi-Scale, Dynamic Recycle Inflow Condition Approach , LUCIANO CASTILLO, Rensselaer Polytechnic Institute, JUAN G. ARAYA, Swansea University Swansea, CHARLES MENEVEAU, The Johns Hopkins University, KENNETH JANSEN, Rensselaer Polytechnic Institute — A method for prescribing realistic turbulent velocity inlet boundary conditions is presented for simulations of spatially evolving turbulent boundary layers. The standard rescaling process requires prior knowledge about how the appropriate velocity and length scales are related between the inlet and recycle stations. In the present study the scales for the inner and outer regions are determined from the multi-scale approach based on the original equilibrium similarity method developed by Castillo and George (2001) (for PG flows). In addition, a new dynamic approach is proposed in which power law ratios of inner/outer scales are used with scaling exponents that may depend on flow conditions and are deduced dynamically by involving an additional plane, a "test plane". This improvement, as well as the use of multiple velocity scales, permits the simulations of turbulent boundary layers subjected to arbitrary pressure gradients. DNS for zero (ZPG), and pressure gradient flows (APG and FPG) are discussed with special emphasis on adverse pressure gradient flows. In addition, new simulations at about $R_\theta \simeq$ 3,000 will be shown and compared with experimental data.

11:35AM BA.00006 Effects of Mach Number on Near-Wall Turbulence Structures in Supersonic Turbulent Boundary Layers[1] , MAHER LAGHA, JOHN KIM, JEFF ELDREDGE, XIAOLIN ZHONG, UCLA — In order to examine the effects of high Mach number on turbulence structures in compressible turbulent boundary layers, we performed direct numerical simulation of a spatially evolving supersonic turbulent boundary layer. A hybrid numerical method coupling the 5th-order WENO scheme with a 5th-order upwind finite-difference scheme was used in order to compute turbulence accurately while capturing sharp gradients that might exist in high Mach number flows. The rescale-and-recycle method was used for the inflow boundary condition. The Mach numbers were varied from 2.5 to 7 while the Reynolds numbers based on the boundary-layer displacement, wall-shear velocity and the viscosity at the wall were kept about the same. In order to determine the relative importance of the true compressibility effect and the variable-property effect in high Mach number boundary layers, we also solved modified Navier-Stokes equations, in which the energy equation was modified to include a heat sink term (Coleman, Kim and Moser, J. Fluid Mech. 305). The effects of Mach number on turbulence statistics as well as an analysis of the near-wall turbulence structures will be presented.

[1]This work is supported by the NASA Grant/Cooperative Agreement NNX08AB39A . Monitor. Alan Wray of NASA Ames Research Center.

11:48AM BA.00007 Direct numerical simulations of hypersonic boundary layers varying wall-to-freestream temperature ratio[1] , LIAN DUAN, IZAAK BEEKMAN, PINO MARTIN, Princeton University — The effects of wall-temperature condition on the statistics of compressible turbulent boundary layers are investigated using direct numerical simulation (DNS). DNS of turbulent boundary layers at Mach 5 with the ratio of wall-to-edge temperature Tw/Te from 1.0 to 5.4 (Cases M5T1 through M5T5) are performed. Case M5T5 corresponds to nearly adiabatic wall, and cases M5T1 through M5T4 correspond to cooled isothermal walls. The validity of Morkovin's scaling, Walz's equation, and the standard and modified strong Reynolds analogy are assessed. Turbulent kinetic energy, contours of spanwise vorticity, near-wall streaks, and two-point correlations show that the temperature cooling stabilizes the turbulence in compressible boundary layers. Compressibility effects are enhanced by wall cooling but remain secondary to the dynamics already appearing in incompressible flow, and the turbulence dissipation remains primarily solenoidal. Hairpin packets are stronger and more coherent for colder wall simulations, while the average hairpin angle remains insensitive to wall temperature.

[1]Funded by NASA Grant # NNX08AD04A and NASA Grant # NCC3-989.

12:01PM BA.00008 Large eddy simulation of boundary layers with embedded spanwise vortices , IFTEKHAR NAQAVI, UGO PIOMELLI, Queen's University — We performed large eddy simulations of a zero-pressure-gradient boundary layer interacting with strong, spanwise-oriented vortices. A periodic array of spanwise vortices is generated at the edge of the boundary layer; as they advect downstream, they introduce strong perturbations that extend to the near-wall region, and alter significantly the turbulence dynamics. Localized separation regions are observed below the vortices. Phase averaged data show that the vortices have lost their coherence 10δ downstream of the point where they are generated however the boundary layer does not recover its equilibrium state even after 60δ. While the vortices remain coherent the phase-averaged velocity profiles show a strong wake, and the logarithmic law is shifted first upwards, and then downwards as the vortex passes. After the coherence is lost all the phases show almost identical behaviour, but the recovery towards an equilibrium boundary layer profile is slow. The fluctuations due to the vortices can be decomposed into a periodic and a random component. The periodic fluctuations supply up to 50% of the total Reynolds stresses in the coherent region, but their contribution is reduced significantly when the vortices decay, and the Reynolds stress due to the random fluctuations is dominant. Flow visualizations show strong uplift and stretching of the near-wall vortices.

12:14PM BA.00009 Large-eddy simulation investigation of large-scale structures in a long channel flow[1] , DANIEL CHUNG, BEVERLEY MCKEON, California Institute of Technology — We report statistics of large-scale near-wall structures from large-eddy simulation (LES) of turbulent channel flow at friction Reynolds numbers 2 k and 200 k. To properly assess the behavior of large-scale structures, we perform simulations in a channel whose length is 96 times its half-channel width. In agreement with experiments, these large-scale structures were found to modulate the statistics of the underlying small-scale fluctuations. In particular we report that, near the wall, large-scale high-speed streaks carry more intense superimposed small-scale fluctuations, but that this correlation is reversed away from the wall. We also report that the convection velocity of these large-scales near the wall departs slightly, but unequivocally, from the mean velocity.

[1]Funded by NSF.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BB Turbulent Mixing 101B

10:30AM BB.00001 Turbulent transport of a high Schmidt number passive scalar below an air–water interface with zero mean shear[1] , EVAN VARIANO, UC Berkeley, EDWIN COWEN, Cornell University — Laboratory measurements in a stirred tank reveal the turbulent flux on the water side of an air–water interface. The tank is stirred from below by random jets, providing turbulence that is homogeneous and isotropic in the horizontal direction. This flow is an interesting counterpoint to cases where the turbulence at the surface is driven by shear there. The Taylor microscale Reynolds number $R_\lambda \approx 300$, giving a wide range of scales. The contribution of these various scales to turbulent scalar flux is investigated by simultaneously measuring the velocity field and the concentration field of dissolved CO_2. This is accomplished via quantitative imaging, and allows cospectra, structure functions, and coherent structures to be investigated. These are compared to theory for sheared interfaces, as well as to similar results obtained by other researchers at lower Schmidt and Reynolds numbers via CFD.

[1]Supported by National Science Foundation (Grant No. CTS-0093794)

10:43AM BB.00002 Effect of the initial conditions on scalar dispersion in uniform-density grid turbulence[1] , JEAN-FRANCOIS KRAWCZYNSKI, DANIEL LANG, PAUL DIMOTAKIS, California Institute of Technology — According to the equilibrium similarity analysis, self-preservation solutions are valid at all scales of motion for many 'classical' flows. In grid-turbulence for example, the energy decay rate is of a power-law form and can depend on initial conditions, hence decay-rate constants cannot be universal, except possibly in the limit of "infinite" Reynolds number. Most of the earlier attempts to validate this theory do not cover a sufficiently wide range of initial conditions or decay times. In the present work, a passive scalar is released from a point source in uniform-density grid turbulence (downstream of a towed grid) using three different momentum-flux conditions: 1) a momentum-less-wake condition, 2) a wake condition, and 3) a jet condition. Using laser-induced fluorescence techniques, data obtained from stream-wise cuts of the scalar field are recorded for the complete turbulence decay range, with varying initial Reynolds numbers based on the mesh length of the grid reaching $\mathrm{Re}_M = 5.6\times10^4$. Using fitting parameters to extract the power-law decay range, the scaling for the different inflow conditions is computed and differences are discussed. It is shown that scalar dispersion in grid turbulence retains the influence of the initial conditions over the entire decay range.

[1]Supported by AFOSR.

10:56AM BB.00003 Effect of scalar-field initial conditions on the internal intermittency of turbulent passive scalars[1] , JASON LEPORE, LAURENT MYDLARSKI, McGill University — Lepore and Mydlarski[2] recently measured the inertial-convective-range structure function scaling exponents of a turbulent passive scalar field (ξ_n) in hydrodynamically identical flows for which the passive scalar field was injected using two different techniques. The flow under consideration was the turbulent wake of a circular cylinder and the scalar fields were injected by: (i) heating the cylinder, and (ii) use of a mandoline. Lepore and Mydlarski showed that the higher-order structure function scaling exponents were dependent on the passive scalar field boundary conditions – a result that (i) explained the previously observed variations in ξ_n reported by different research groups (who generated their passive scalar fields using different methods), and (ii) is, ostensibly, in violation of Kolmogorov-Oboukhov-Corrsin theory. The present work further explores the role of the passive scalar field boundary conditions in turbulent mixing by means of additional statistical analyses (conditional expectations, inverse structure functions of passive scalar increments, etc.) in an attempt to elucidate the relationship between the boundary conditions and the internal intermittency of the turbulent passive scalar field.

[1]Support has been graciously provided by the NSERC (Canada).
[2]Lepore and Mydlarski, 2009, *Phys. Rev. Lett.*, **103**, 034501.

11:09AM BB.00004 On the universality of dissipation mechanisms in mixing , ANDREW LAWRIE, STUART DALZIEL, University of Cambridge — We consider partitioning of energy in incompressible miscible variable density flows and in particular choose to distinguish between energy that is dissipated due to viscosity, and that which performs mixing. Variable density fluids exhibit a change in potential energy exclusively associated with mixing, as well as an overall change due to the system relaxation. The relationship between these energies provides a global measure of mixing efficiency that can be easily diagnosed from experiments and comparable simulations. Rayleigh-Taylor instability is one of the most efficient mechanisms for mixing: typically half of the initially available potential energy is used in doing so, an efficiency of 50%. Unfortunately initial and final states in a Rayleigh-Taylor unstable flow are such that, by construction, mixing efficiency is bounded at 50%. Our new configuration has an energetically admissible end-state that could be achieved if the mixing efficiency were 75%, but our experiments show that despite the possibility of achieving a higher mixing efficiency, the system once more relaxes to 50%. We conclude that the processes governing the distribution of energy are micro-scale, and not pre-determined by initial conditions.

11:22AM BB.00005 On the diapycnal diffusivity in homogeneous stably stratified turbulence , DEREK STRETCH, University of KwaZulu-Natal, SUBHAS VENAYAGAMOORTHY, Colorado State University — Quantifying the irreversible diapycnal mixing that occurs in stably stratified turbulence is fundamental to the understanding and modeling of geophysical flows, and for predicting dispersion in these flows. In this study, data of diapycnal mixing from direct numerical simulations of homogeneous stably stratified turbulence, both with and without shear, and from grid turbulence experiments, are reviewed and analyzed to investigate the scaling of the diapycnal diffusivity. In these homogeneous flows the instantaneous diapycnal diffusivity is given exactly by $K_d = \epsilon_\rho / \left(\partial\overline{\rho}/\partial z\right)^2$ and may be expressed in terms of the large scale properties of the turbulence as $K_d = \gamma L_E^2/T_L$, where $L_E = (\overline{\rho'^2})^{\frac{1}{2}}/|\partial\overline{\rho}/\partial z|$ is the Ellison overturning length-scale, $T_L = k/\epsilon$ is the turbulence decay time-scale, and γ is half the mechanical to scalar time-scale ratio. Our results show that L_E and T_L can explain all the variations in K_d over a wide range of shear and stratification strengths (including shear-free and neutrally stratified cases) while γ remains approximately constant. This result is also found to be independent of Prandtl (or Schmidt) number.

11:35AM BB.00006 The dispersion of patterns written in turbulent air , MEHRNOOSH MIRZAEI, NICO DAM, HANS TERMEULEN, Institute of Molecules and Materials, Applied Molecular Physics, Radboud University, Nijmegen, the Netherlands, WILLEM VAN DE WATER, Physics Department, Eindhoven University of Technology, PO Box 513, 5600 MB Eindhoven, the Netherlands — We study the mixing of passive objects in turbulence by writing structures in turbulent air and following their deformation in time. The writing is done by fusing O_2 and N_2 molecules into NO in the focus of a strong ultraviolet laser beam. By crossing several of these laser beams, patterns that have both small and large scales can be painted. The patterns are visualized a while later by inducing fluorescence of the NO molecules with a second UV laser and registering the image. The width of the lines that make the pattern is approximately $50\,\mu$m, a few times the Kolmogorov length, the overall size of the patterns (≈ 4mm) is inside the inertial range of the used turbulent jet flow. Thus, we are able to study turbulent dispersion both at micro- and macroscales in a frame of reference that moves with the flow. In this way we have measured the spreading of clouds whose size is a few times the Kolmogorov length and the Batchelor dispersion of objects whose size is inside the inertial range. Patterns are compressible objects and spontaneously develop concentration fluctuations. We show for the first time the remarkable statistical properties of these fluctuations.

11:48AM BB.00007 Enthalpy Diffusion in Multicomponent Flows[1] , ANDREW COOK, LLNL — The enthalpy diffusion flux in the multicomponent energy equation is a well known yet frequently neglected term. It accounts for energy changes associated with compositional changes resulting from species diffusion. The term prevents local violations of the entropy condition in flows where significant mixing occurs between species of dissimilar molecular weight. In simulations of nonpremixed combustion, omission of the enthalpy flux can lead to anomalous temperature gradients, which may cause mixing regions to exceed ignition conditions. The term can also play a role in generating acoustic noise in turbulent mixing layers. Euler solvers that rely on numerical diffusion to blend fluids at the grid scale cannot reliably predict temperatures in mixing regions. On the other hand, Navier-Stokes solvers that incorporate enthalpy diffusion can provide much more accurate results. In constructing turbulence closures for high Reynolds number mixing, the same turbulent diffusion model that appears in the species mass transport equation should also appear in the energy equation as part of a "turbulent enthalpy diffusion"; otherwise the energy and species transport equations will not be consistent.

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

12:01PM BB.00008 A bivariate beta distribution as a presumed pdf for two mixture fractions , ERIC DORAN, HEINZ PITSCH, Stanford University — In turbulent reacting flows, an assumed pdf of a conserved scalar is often used to characterize the composition of the fluid. If more than a single fuel stream is present multiple mixture fractions are necessary, in which case a representation of the joint statistics is required. Here, several joint distributions proposed in the literature and a new bivariate beta distribution are investigated with regard to their suitability as a presumed pdf for the mixing of two conserved scalars. The bivariate beta distribution has the advantage that the marginal distributions reduce to univariate beta distributions, which have been shown to perform well for the mixing of a single conserved scalar. The presumed distributions are compared to DNS data over a range of initial scalar fields, including variations in the state of mixing and proportions of each scalar.

12:14PM BB.00009 Numerical Implementation of Molecular Transport and Mixing in LES/FDF of Turbulent Flows , SHARADHA VISWANATHAN, STEPHEN POPE — In large-eddy simulations of turbulent flames, the effect of molecular diffusion on scalar transport is significant and therefore needs to be modeled correctly in Lagrangian Filtered Density Function (FDF) methods. McDermott *et al.* (2007) show that in FDF methods, modeling molecular diffusion as a mean drift term in the scalar equation avoids the spurious production of scalar variance. Following McDermott *et al.*, we incorporate the effects of molecular diffusion in the Interaction by Exchange with the Mean (IEM) mixing model. In this study, we evaluate two second-order-accurate numerical methods *viz.* Particle-In-Cell (PIC) and Cloud-In-Cell (CIC) for implementing mixing. Given the nominal particle number density and problem geometry in a typical FDF calculation of a jet flame, the estimated mean field needs to be smoothed both for variance reduction and for proper treatment of empty cells near the jet axis. Our numerical studies show that while both implementations achieve detailed conservation and guarantee boundedness of the scalar field, CIC is computationally expensive and PIC is dependent on the scalar bounds. But for a given accuracy, PIC with smoothing incurs typically half the cost of CIC, for an appropriate choice of the smoothing parameter. This research is supported by the Department of Energy under Grant No. DE-FG02-90ER.

12:27PM BB.00010 Large-Eddy simulation and measurements of turbulent mixing in a confined rectangular jet , JAMES HILL, BO KONG, MICHAEL OLSEN, RODNEY FOX, Iowa State University — Large-eddy simulations (LES) of a passive scalar were performed for a confined rectangular liquid jet (Re = 20,000) and compared with the simultaneous particle image velocimetry (PIV) and planar laser induced fluorescence (PLIF) measurements. A finite-difference LES code was used to obtain velocity data, which was already proved to provide a very good agreement with PIV experiment data in the previous study. Both finite-difference and finite-volume formulation were used to discretize and solve the filtered scalar transport equation. The effects of numerical schemes and subgrid models on the LES results were investigated. Model validation was performed by comparing LES data for one-point statistics such as the passive scalar mean and variance, turbulence flux and probability distribution function with the PLIF data. In addition, LES data for the two-point spatial auto-correlations of passive scalar fluctuations and cross-corelations of passive scalar fluctuation and velocity fluctuations were also computed and compared with the simultaneous PIV and PLIF data.

Sunday, November 22, 2009 10:30AM - 12:14PM –
Session BC Turbulence Simulations II 101C

10:30AM BC.00001 Subgrid scale physics in a turbulent boundary layer flow under varying convective stability conditions: an a-priori study , ELIE BOU-ZEID, Princeton University, NIKKI VERCAUTEREN, CHAD HIGGINS, HENDRIK HUWALD, MARC B. PARLANGE, Ecole Polytechnique Federale de Lausanne-EPFL, CHARLES MENEVEAU, Johns Hopkins University — Using data sets collected during the Lake Atmosphere Turbulent Exchanges (LATEX, convectively unstable conditions) and the Snow Horizontal Array Turbulence Study (SnoHATS, convectively stable conditions) field experimental campaigns, we study the impact of this convective stability on the physics of small scale turbulence in the atmospheric boundary layer flow and the implications for modeling the subgrid scales stresses and fluxes (of heat and moisture) in large eddy simulation. Results indicate that the subgrid scale turbulent Prandtl number increases significantly as the flow transitions from unstable to stable. Under all stabilities, the TKE and scalar variance dissipation estimated based on the structure functions are very good estimates of the flux of energy to the subgrid scales; however, under stable conditions, a significant fraction of the TKE flux is destroyed by buoyancy rather than by viscous dissipation. Finally, the effect of stability on the coefficients of 2 SGS models is shown to be better explained by the Ozmidov scale under stable conditions. Overall, these results indicate that subgrid scale modeling is not drastically affected by atmospheric stability and hence a unified approach is possible.

10:43AM BC.00002 A Subgrid Scale Estimation Model for Large Eddy Simulation[1] , RAJES SAU, KRISHNAN MAHESH, University of Minnesota — We discuss a novel estimation procedure to model the subgrid velocity for Large Eddy Simulation. The subgrid stress is obtained directly from the estimated subgrid velocity. The subgrid velocity is modeled as a function of resolved velocity ($\bar{u}_i$) and resolved strain–rate tensor ($\bar{S}_{ij}$). Using tensor invariants, we obtain an expression for subgrid velocity involving $\bar{u}_i$, that is quadratic in $\bar{S}_{ij}$ with three undetermined coefficients. These three coefficients are obtained by imposing the following constraints: (i) Galilean invariance, (ii) ensemble-averaged subgrid dissipation and (iii) local subgrid kinetic energy. Subgrid dissipation is obtained through a new dynamic procedure which uses two scalar level identities without least squares minimization, as opposed to the tensor level Germano identity. Subgrid kinetic energy is obtained either from the dynamic Yoshizawa model or a transport equation for subgrid kinetic energy. The estimation model is applied to isotropic turbulence and good results are obtained. Realistic backscatter is also predicted using this model.

[1]This work is supported by the Air Force Office of Scientific Research (AFOSR) under grant FA-9550-04-1-0064.

10:56AM BC.00003 Grid-independent LES[1] , SANJEEB BOSE, PARVIZ MOIN, Center for Turbulence Research, Stanford University — Grid independent turbulent statistics are obtained in a planar channel flow at $Re_\tau = 640$ by explicit filtering the governing equations for LES. Three dimensional filters (Vasilyev et al., JCP, 1998) are utilized such that commutation error is the same order as the truncation error of the fourth-order, conservative finite difference scheme (Morinishi et al., JCP, 1998). Several calculations are performed with a fixed filter width, but with varying grid resolutions. The grid-independent mean velocity profile is in good agreement with the experimental data of Hussain & Reynolds (1970). The rms velocity profiles and one-dimensional energy spectra are compared with previous LES results and the unfiltered DNS of Abe et al. (2001), and show convergence toward a grid-independent profile. Ensemble averaged contributions of the dynamic Smagorinsky subgrid model to the Reynolds shear stress have also converged to a grid-independent profile across all grid resolutions. The effect of effective filter shape on the convergence of turbulence statistics will be discussed. Progress in the development of grid-independent LES for complex geometries with unstructured meshes will be presented.

[1]Supported by the DOE PSAAP program, the DOE CSGF, and the Stanford Graduate Fellowship.

11:09AM BC.00004 A dynamic wall model constrained by RANS Reynolds stress[1] , AMAN VERMA, NOMA PARK, KRISHNAN MAHESH — We 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. The proposed model is 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. The model has been implemented in spectral and structured finite volume solvers and is being extended to an unstructured solver. These developments will be discussed.

[1]This work was supported by the United States Office of Naval Research under ONR Grant N00014-05-1-0003.

11:22AM BC.00005 LES of Supersonic Turbulent Flows with the Scalar FMDF , ARAZ BANAEIZADEH, ZHAORUI LI, FARHAD JABERI, Michigan State University — The scalar filtered mass density function (FMDF) subgrid-scale model is further developed and tested for large eddy simulation (LES) of supersonic turbulent mixing and reacting flows in complex geometries. The LES/FMDF is implemented via a hybrid numerical method. In this method, the filtered compressible Navier-Stokes equations in curvilinear coordinate systems are solved with a generalized, high-order, multi-block, compact differencing scheme. To reduce the numerical oscillations of the compact scheme in shock regions, a localized high order artificial viscosity is added. The compressible scalar FMDF equation is solved with a stochastic Lagrangian Monte Carlo method. The results obtained with the LES/FMDF for shock tube and other compressible flows indicate that the pressure effects on the scalar field are well captured by the extended compressible FMDF model. The consistency of the filtered temperature and density fields as obtained from the Eulerian (finite difference) and Lagrangian (Monte Carlo) components of the LES/FMDF also indicate the reliability and the accuracy of the model in high speed flows.

11:35AM BC.00006 Dynamic k-Equation Model for LES of Compressible Flows[1] , XIAOCHUAN CHAI, KRISHNAN MAHESH, University of Minnesota — The sub-grid scale (SGS) kinetic energy (KE) has to be modeled in LES of compressible flows. Standard compressible versions of the dynamic Smagorinsky model (DSM) use Yoshizawa's expression for SGS KE. However, it is well known that Yoshizawa's Model tends to under-predict the magnitude of SGS KE. Obtaining the SGS KE from its transport equation,has shown improved performance for incompressible flows (e.g. Ghosal *et al.* 1995, Kim & Menon 1996). We develop a compressible version of the DSM model with SGS KE equation. The SGS KE transport equation for compressible flow is derived, and the unclosed terms in the compressible KE equation are modeled and dynamically closed using the Germano identity. The proposed model is applied to decaying isotropic turbulence and normal shock/isotropic turbulence interaction.

[1]This work is supported by the National Science Foundation under grant NSF-0828162.

11:48AM BC.00007 Subgrid scale modeling approaches for LES of low Mach number Jets , GREGORY RODEBAUGH, LESTER SU, Johns Hopkins University — Large-eddy simulation of the low Mach number equations provides a computationally attractive option for simulating low speed, complex flows with large density and temperature gradients, particularly for reacting flow applications. Accurately predicting the scalar concentration fields is essential for the development of precise combustion simulations. In this work, we seek to elucidate what effects different subgrid scale (SGS) stress and scalar flux models have on both the mixing properties and turbulent statistics of the flow. Additionally, we aim to understand the coupling between the SGS stress and scalar flux models; therefore, we restrict the study to an isothermal flow with two species of different densities. There have been previous investigations of SGS models in compressible flows, but these focused primarily on higher Mach number regimes and SGS models for the energy equation. We study a canonical axisymmetric turbulent jet in this work, with the low Mach number equations being discretized in cylindrical coordinates. A predictor-corrector scheme is employed for time evolution of mass species fraction and momentum.

12:01PM BC.00008 The role of subgrid-scale models near the turbulent/nonturbulent interface in free shear flows , CARLOS B. DA SILVA, IDMEC/IST Technical University of Lisbon — This work addresses a challenging new problem for large-eddy simulations (LES) that exists in free shear flows where there are two distinct 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/NT) interface. It has been shown that important Reynolds stresses exist near the T/NT interface and that these stresses determine in part the mixing and combustion rates in jets. In the present work the role of several subgrid-scale models near the T/NT is analyzed in detail by using direct numerical simulations (DNS) and LES. The subgrid scales of motion near the T/NT interface are far from equilibrium and contain an important fraction of the total kinetic energy. Model constants used in several subgrid-scale models such as the Smagorinsky and the gradient models need to be corrected near the jet edge. The procedure used to obtain the dynamic Smagorinsky constant is not able to cope with the intermittent nature of this region. Both a-priori tests and LES show that near the jet edge the Smagorinsky model is superior both to the dynamic Smagorinsky and to the gradient models.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BD Flow Control II 101D

10:30AM BD.00001 The breakdown of the viscous regime in riblets[1] , RICARDO GARCÍA-MAYORAL, U. Politécnica Madrid, JAVIER JIMÉNEZ, U. Politécnica Madrid and CTR Stanford — We investigate the mechanisms involved in the breakdown of the viscous regime in riblets, with a view to determining the point of optimum performance, where drag reduction ceases to be proportional to the riblet size. This occurs empirically for a groove cross-section $A_g^+ \approx 120^+$. To study the interaction of the riblets with the overlaying turbulent flow, we systematically conduct DNSes in a ribbed turbulent channel with increasing riblet size. The conditionally averaged crossflow above and within the grooves reveals a mean recirculation bubble that exists up to the point of viscous breakdown, isolating the groove floor from the overlying crossflow, and preventing the high momentum fluid from entering the grooves. We do not find evidence of outside vortices lodging within the grooves until $A_g^+ \approx 400$, which is well past the drag minimum, and already into the drag increasing regime. Interestingly, as the bubble breaks down, we observe that quasi-two-dimensional spanwise structures form just above the riblets, similar to those observed above porous surfaces and plant canopies, which appear to be involved in the performance degradation.

[1]Funded by AVERT program of the EC, and CICYT.

10:43AM BD.00002 Linear Control of Turbulent Channel Flow and the Role of Pressure , JONATHAN MORRISON, ATI SHARMA, Imperial College, BEVERLEY MCKEON, California Institute Technology — The response of fully turbulent channel flow to global, linear control is examined. Through full-domain sensing on the wall- normal velocity, control is designed to ensure that the perturbations decay monotonically. The physical effect is such that vdU/dy is countered directly. The control is shown to work for flow disturbances of any size and not those small enough to permit linear approximation. When the forcing bandwidth is progressively reduced, control in terms of drag reduction remains effective. The response of the near-wall flow at $y^+ = 15$ to full-domain actuation is examined in detail, with particular emphasis on the pressure field. It is shown that the near-wall pressure fluctuations are attenuated more quickly than those associated with both the velocity and vorticity fields. Reasons for this result are examined, in particular the pressure- gradient fluctuations, which drive the momentum field, are examined.

10:56AM BD.00003 Control of transition in Poiseuille flow using streamwise traveling waves. Part 1: Receptivity analysis[1] , RASHAD MOARREF, MIHAILO R. JOVANOVIĆ, University of Minnesota — We assess the efficacy of a zero-net-mass-flux blowing and suction in the form of streamwise traveling waves for transition control in Poiseuille flow. As shown by Bewley (2009), the net efficiency is always negative if the uncontrolled flow stays laminar. We show, however, that a positive net efficiency can be achieved in situations where the uncontrolled flow becomes turbulent but the controlled flow remains laminar. Starting from this observation, we develop a framework for selection of traveling wave parameters for control of transition with a positive net power balance. Our detailed parametric study shows that, relative to the uncontrolled flow, the velocity fluctuations around the upstream traveling waves at best exhibit similar receptivity to background disturbances. In contrast, the properly designed downstream traveling waves can significantly reduce receptivity which makes them well-suited for preventing transition. Our theoretical predictions are confirmed by direct numerical simulations of the Navier-Stokes equations in Part 2 of this paper.

[1]National Science Foundation (under CAREER Award CMMI-06-44793) and University of Minnesota Supercomputing Institute.

11:09AM BD.00004 Control of transition in Poiseuille flow using streamwise traveling waves. Part 2: Direct numerical simulations[1] , BINH LIEU, RASHAD MOARREF, MIHAILO R. JOVANOVIĆ, University of Minnesota — This work builds on and confirms the theoretical findings of Part 1 of this paper. We use direct numerical simulations (DNS) of the Navier-Stokes (NS) equations to assess the efficacy of blowing and suction in the form of streamwise traveling waves for transition control in Poiseuille flow. We highlight the effects of the modified base flow on the dynamics of velocity fluctuations and the net efficiency. Our simulations verify theoretical predictions of Part 1 that the upstream traveling waves promote turbulence even when the uncontrolled flow stays laminar. On the other hand, the downstream traveling waves with parameters selected in Part 1 are capable of reducing the fluctuations' kinetic energy and maintaining the laminar flow. For this choice of control, a positive net efficiency of $26\,\%$ compared to the turbulent uncontrolled flow can be achieved. The DNS results of this paper elucidate the predictive power of the method developed in Part 1 and suggest that the linearized NS equations with uncertainty may serve as an effective control-oriented model for preventing transition.

[1]National Science Foundation (under CAREER Award CMMI-06-44793) and University of Minnesota Supercomputing Institute.

11:22AM BD.00005 Shape Optimization of Peristaltic Pumping[1] , SHAWN WALKER, MICHAEL SHELLEY, New York University — Transport is a fundamental aspect of biology and peristaltic pumping is a fundamental mechanism to accomplish this; it is also important in many industrial processes. We present a variational method for optimizing peristaltic pumping in a two dimensional periodic channel with moving walls to pump fluid. No a priori assumption is made on the wall motion, except that the shape is static in a moving wave frame. Thus, we pose an infinite dimensional optimization problem and solve it with finite elements. Sensitivities of the cost and constraints are computed variationally via shape differential calculus and L^2-type projections are used to compute quantities such as curvature and boundary stresses. Our Optimization method falls under the category of sequential quadratic programming (SQP) methods. As a result, we find optimized shapes that are not obvious and have not been previously reported in the peristaltic pumping literature. Specifically, we see highly asymmetric wave shapes that are far from being sine waves. Many examples are shown for a range of fluxes and Reynolds numbers up to $\mathrm{Re} = 500$ which illustrate the capabilities of our method.

[1]This work was supported by the NSF and DOE.

11:35AM BD.00006 Computational Sensitivity Analysis of Low-Reynolds Number Turbulent Channel Flow[1] , RICHARD KIRKMAN, MEREDITH METZGER, University of Utah — Computational sensitivity analysis has been performed for low-Reynolds number turbulent flow in a plane channel. Two methods: (i) the continuous sensitivity equation method (CSEM) and (ii) complex step differentiation (CS), have been implemented in the context of direct numerical simulations to determine the sensitivity derivatives (or coefficients) of the primitive variables to changes in the Reynolds number. Simulations were performed at Reynolds numbers of 100 and 180, based on the friction velocity and channel half-width. Turbulent velocity statistics compare very well to others in the literature (Kim et al., 1987; Kuroda et al., 1989). The sensitivity results correctly predict the expected change in both the mean streamwise velocity and Reynolds shear stress profiles to changes in Reynolds number. Furthermore, the mean sensitivity results correctly predict the local slope of the skin friction coefficient versus Reynolds number. The *instantaneous* sensitivity results also reveal that regions of high magnitude sensitivity correlate to regions containing coherent structures in turbulent channel flow. The additional computational expenses incurred in order to run the computational sensitivity simulations in this context are also discussed.

[1]Supported by the National Science Foundation (IIS-0428856).

11:48AM BD.00007 Prediction and Manipulation of Skin Friction in High-Reynolds Number Flows[1] , YULIA PEET, Northwestern University, PIERRE SAGAUT, University of Pierre and Marie Curie - Paris 6 — Skin friction, which depends on the local shear rate, becomes difficult to predict when Reynolds number of the flow exceeds its critical value, and the flow becomes disorderly. We have developed an analytical approach which relates skin friction coefficient to statistical information in the flow above the surface in a general case of high-Reynolds numbers and complex wall shapes. Current approach allows separating the contribution of different dynamical effects into skin friction, thus providing the basis for understanding how skin friction can be manipulated by passive and active flow control methods. In the current presentation, we show the effects of organized surface nonuniformities on skin friction and compare the results of theoretical analysis with numerical simulations.

[1]Support of Agence Nationale pour la Recherche (ANR) of France is acknowledged.

12:01PM BD.00008 Hybrid Manipulation of Streamwise Vorticity in a Turbulent Boundary Layer[1] , ABRAHAM N. GISSEN, BOJAN VUKASINOVIC, ARI GLEZER, Georgia Tech — Manipulation of streamwise vorticity in a turbulent boundary layer is investigated experimentally at high subsonic speeds ($M = 0.5$) along converging-diverging duct wall designed to provide an adverse pressure gradient that mimics the pressure gradient in a typical offset diffuser. Counter-rotating vortex pairs and single-sense vortices are formed and characterized using conventional passive sub-boundary layer micro-ramps and micro-vanes, respectively. Fluidic analogues of these passive devices are established by using surface-mounted synthetic jet actuators. Hybrid manipulation of streamwise vorticity within the boundary layer is demonstrated by simultaneous combination of passive and active actuation which enables robust, controllable "fail-safe" operation that requires no net mass injection.

[1]Supported by NASA and the Boeing Company.

12:14PM BD.00009 ABSTRACT WITHDRAWN –

12:27PM BD.00010 Spanwise Varying Open-Loop Control of a Backward-Facing Step Flow[1] , AARON BAUGH, STUART GILBERT, MARC SCHOSTEK, DAVID BREAKEY, LORENZ SIGURDSON, Vortex Fluid Dynamics Lab, University of Alberta — We are experimentally investigating active control of the reattaching shear layer downstream of a backward-facing step in water. Transitional and turbulent separation bubbles are studied. Control is achieved using 128 hydraulic suction-and-blowing actuation ports along the span at the corner of the step where the flow first separates, inspired by Sakakibara and Anzai's design for a plane jet [*Phys. Fluids* 13, 1541 (2001)]. Perturbation magnitudes vary in space across the span of the step, and each port's perturbation is periodic with zero-net mass flux. This actuation technique is the physical manifestation of some of the numerical simulations of Kang and Choi [*J. Fluid Mech.* 463, 201 (2002)]. We use backlit dye to track the evolution of vorticity in the reattaching shear layer. Tufts, which have been more commonly employed in aerodynamic studies, are adapted here for use in water. An array of approximately 1000 tufts is in place downstream of the step to examine the effects of the control schemes on the length of the recirculation bubble.

[1]We gratefully acknowledge NSERC Discovery Grant 41747.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BE Biofluids II: General II - Biologically Generated Flows 101E

10:30AM BE.00001 Experiments on blood-sucking mechanism of a female mosquito[1] , SANG JOON LEE, Dept. of Mech. Eng., POSTECH, Republic of Korea, BO HEUM KIM, I-Bio, POSTECH, JUNG YEOP LEE, Dept. of Mech. Eng., POSTECH, BIOFLUID AND BIOMIMIC RESEARCH CENTER TEAM — The blood-sucking phenomena of a female mosquito were investigated experimentally At first, the velocity fields of blood-sucking flow inside the proboscis of a female mosquito were measured consecutively using a micro particle image velocimetry (PIV) system The velocity signals of the blood-sucking flow in the proboscis show a periodic pulsatile flow pattern and the spectral analysis of the velocity waveform exhibits a clear peak at 6.1 Hz. The blood flow inside the proboscis has a parabolic profile, similar to that of a Hagen-Poiseuille flow. In addition, the synchrotron X-ray micro-imaging technique was employed to visualize the dynamic movement of the two pumping organs (cibarial pump and pharyngeal pump) inside the head of blood-sucking using iodine solution as a contrast material. The temporal variation of the two pump organs of a female mosquito was found to be superior, compared to that of a male mosquito. In addition, we found the functional relationship of the two pumps operating in a systematic manner with a small phase difference.

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

10:43AM BE.00002 Microscopic filter feeders near boundaries , RACHEL PEPPER, Harvard University, MARCUS ROPER, University of California, Berkely, SANGJIN RYU, Massachusetts Institutte of Technology, PAUL MATSUDIARA, National University of Singapore, HOWARD STONE, Princeton University — We show through calculations, simulations, and experiments that the eddies often observed near sessile filter feeders are due to the presence of nearby boundaries. We model the common filter feeder *Vorticella*, which is approx 50 μm across and which feeds by removing bacteria from ocean or pond water that it draws towards itself. We use an analytic stokeslet model and a Brinkman flow approximation with the organism modeled as a cylinder with two different boundary conditions to predict the size of the eddy caused by two parallel no-slip boundaries that represent the slides between which experimental observations are often made. We also use three-dimensional finite-element simulations to fully solve for the flow around a model *Vorticella*. Additionally, we track particles around live feeding *Vorticella* in order to determine the experimental flow field. Our models are in good agreement both with each other and with the experiments. We also show through calculations that filter feeders such as *Vorticella* can greatly enhance their nutrient uptake by feeding at an angle rather than perpendicular to a substrate.

10:56AM BE.00003 Modification of boundary layer momentum by the presence and pumping behavior of the bivalve clam, *Mercenaria mercenaria*, in a tidal channel , S.K. DELAVAN, D.R. WEBSTER, Georgia Tech — The presence and activity of biological organisms have the potential to modify turbulent boundary layer characteristics in natural field settings. To determine the effect of the presence and pumping behavior of the bivalve clam, *Mercenaria mercenaria*, on the boundary layer momentum, profiles were collected for flood tides in the tidal rivers adjacent to Wassaw Sound, Georgia, USA. Velocity profiles were collected simultaneously with two adjacent Acoustic Doppler Velocimeters for boundary layer flows above sediments with and without the presence of buried clams. Treatment sites included clams buried in mud sediments, sand sediments, downstream of oyster beds, and downstream of sea grass beds. We hypothesize that the modification of boundary layer momentum is unique to the treatment characteristics. Vertical profiles of mean velocity, turbulent kinetic energy, and Reynolds shear stress are calculated from the collected time records. Preliminary analysis suggests that flows downstream of sea grass and oyster beds are less affected by the presence of clams than flows over sand and mud flats. Clams reduce the horizontal velocity values above mud substrates when compared to adjacent measurements without clams present, particularly close to the substrate. When buried in sand flats, clams tend to increase the horizontal velocity values higher in the water column.

11:09AM BE.00004 Experimental Investigation of 3-D flow fields around the mouth of the Dwarf Seahorse during attacks on planktonic prey , BRAD GEMMELL, EDWARD BUSKEY, U. Texas at Austin, JIAN SHENG, U. Minnesota — Copepods are an important planktonic food source for fish species. High predation has led to the development of effective escape responses with short reaction times (less than 2 ms), maximum escape velocities of over 500 body lengths per second and shear sensitivity as low as $1.5s^{-1}$. Using high speed digital holography (2 kfps), we measure 3-D distributions of velocity generated by a dwarf seahorse (*Hippocampus zosterae*) during attacks on its copepod prey, *Acartia tonsa*. It is found that successful attacks often produce smaller or even no detectable hydrodynamic disturbances around the strike zone, when compared to unsuccessful attempts. In this paper, we will provide quantitative characterization of this "low-flow" zone. Further, to elucidate the role of a possible geometrical advantage of the seahorse's head in minimizing its bow wave, high-speed time resolved PIV measurements are conducted in a low-speed water tunnel. On-going analysis will provide insights and implications in understanding the dynamics of flows around the stagnation point at high Reynolds number flow. Sponsored by NSF.

11:22AM BE.00005 A Darwinian mechanism for biogenic ocean mixing , KAKANI KATIJA, Bioengineering, California Institute of Technology, JOHN DABIRI, Bioengineering and Aeronautics, California Institute of Technology — Recent observations of biogenic turbulence in the ocean have led to conflicting ideas regarding the contribution of animal swimming to ocean mixing. Previous measurements indicate elevated turbulent dissipation in the vicinity of large populations of planktonic animals swimming in concert. However, elevated turbulent dissipation is by itself insufficient proof of substantial biogenic mixing. We conducted field measurements of mixing efficiency by individual *Mastigias sp.* (a Palauan jellyfish) using a self-contained underwater velocimetry apparatus. These measurements revealed another mechanism that contributes to animal mixing besides wake turbulence. This mechanism was first described by Sir Charles Galton Darwin and is in fact the dominant mechanism of mixing by swimming animals. The efficiency of Darwin's mechanism (or drift) is dependent on animal shape rather than fluid length scale and, unlike turbulent wake mixing, is enhanced by the fluid viscosity. Therefore, it provides a means of biogenic mixing that can be equally effective in small plankton and large mammals.

11:35AM BE.00006 A Lagrangian approach to identifying vortex pinch-off[1] , CLARA O'FARRELL, JOHN O. DABIRI, California Institute of Technology — There exists a physical limit to the size of an axisymmetric vortex ring beyond which it rejects further vorticity flux, and a trailing jet forms behind it. This process is termed "vortex pinch-off," and it has been correlated with maximally efficient fluid transport in pulsed jets[2]. The established method for identifying vortex pinch-off consists of measuring the circulation of the vortex ring after it has separated from its trailing shear layer, and comparing it to the total circulation to determine the instant when the vortex ring ceased to accept vorticity[3]. However, this method relies heavily on the vorticity field, which breaks down due to viscous diffusion in low Reynolds number and unsteady biological flows. We introduce a criterion for identifying pinch-off based on the Lagrangian coherent structures (LCS) in the flow, which is found to be in good agreement with the established criterion based on circulation. The Lagrangian criterion is frame-invariant and does not rely on the vorticity field, and so is a useful tool in the study of complex biological vortex-shedding flows.

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

[2]Krueger and Gharib, Phys. Fluids, **15**, p. 1271, 2003.

[3]Gharib *et al.*, J. Fluid Mech., **360**, p. 121, 1998.

11:48AM BE.00007 A paradox of hovering insects in two-dimensional space[1] , MAKOTO IIMA, Hokkaido University — A paradox concerning the flight of insects in two-dimensional space is identified: insects maintaining their bodies in a particular position (hovering) cannot, on average, generate hydrodynamic force if the induced flow is temporally periodic and converges to rest at infinity. This paradox is derived by using the far-field representation of periodic flow and the generalized Blasius formula, an exact formula for a force that acts on a moving body, based on the incompressible Navier-Stokes equations. This paradox provides insight into the effect of the singular behavior of the flow around hovering insects: the far-field wake covers the whole space.

Reference:
M. Iima, *J. Fluid Mech.*, (2008), **617**, 207–229.

[1]A Grant-in-Aid for Young Scientists (B), 2007-2008, 19740228, and Scientific Research on Priority Areas, 2008-2009, 20033009 from the Ministry of Education, Science, Sports and Culture of Japan.

12:01PM BE.00008 Butterfly proboscis as a biomicrofluidic system , KONSTANTIN KORNEV, Clemson University, DARIA MONAENKOVA, STEVEN REA, CAMPBELL YORE, CALEB KLIPOWICS, KARA EDMOND, VIJOYA SA — It looks amazing how butterflies and moths with their thin feeding trunk are being able to sip very thick liquids like nectar or animal extractions. Their sucking ability goes beyond that: one can observe butterflies and moths probing liquids from porous materials like fruit flesh or wet soils. This suggests that the suction pressure produced by these insects is sufficiently high. The estimates based on engineering hydraulic formulas show that the pressure can be greater than one atmosphere, i.e. it can be greater than that any vacuum pump could supply. In this experimental study, the principles of interfacial flows are used to carefully analyze the feeding mechanism of butterflies and moths. We document the feeding rates and proboscis behavior of Monarch butterflies (Danaus plexippus) in different situations: when butterfly feeds from droplets, from vials modeling floral cavities, and from porous materials modeling fruits, wet soils, or dung. Using high speed imaging and simple models, we propose a scenario of butterfly feeding which is based on capillary action. According to the proposed mechanism, the trunk of butterflies and moths works like a fountain pen where the air bubbles play a significant role in controlling fluid flow.

12:14PM BE.00009 Implicit Large Eddy Simulation of Flow over a Corrugated Dragonfly Wing Using High-Order Spectral Difference Method[1] , Z.J. WANG, Iowa State University — Implicit large eddy simulations of flow over a corrugated dragonfly wing at a Reynolds number of 34,000 at high angles of attack have been investigated with a high-order unstructured spectral difference Navier-Stokes solver. The computational results are compared with a recent experimental study by Hu et al. Both 2D and 3D simulations are carried out to assess how realistic and reliable the 2D simulations are in comparison with 3D simulations. At the angle of attack of 16 degrees, the 2D simulation failed to predict the stall observed in the experiment, while the 3D simulation correctly predicted the stall. In addition, the 3D simulation predicted a mean lift coefficient within 5% of the experimental data. We plan to compute at least another angle of attack and compare with the experimental data. The numerical simulations demonstrated the potential of the high-order SD method in large eddy simulation of physically complex problems.

[1]The research is support by AFOSR and DOE.

12:27PM BE.00010 Effect of flexibility of wing on aerodynamic performance in plunge motion , TUYEN QUANG LE, Konkuk University, Seoul, Korea, JIN HWAN KO, Seoul National University, Seoul, Korea, SOO HYUNG PARK, DOYOUNG BYUN, Konkuk University, Seoul, Korea — The improvement of aerodynamic performance in plunge motion is investigated through a flexibility of chordwise and spanwise directions by applied a prescribed deformation on a airfoil at Reynolds number of 30000. The aerodynamic performance of flapping motion can be evaluated through a thrust force, a input power and a propulsive efficiency which is a ratio of propulsive power which generate from thrust force to input power. Unsteady viscous flows over NACA 0012 airfoils in plunge motions are computed by using a time-accurate preconditioned Navier-Stokes solver coupled with a volume grid deformation code. For spanwise or chordwise flexibility, there is a optimal phase angle for maximum thrust force and propulsive efficiency. Especially, a combination of spanwise and chordwise flexibility can improve aerodynamic performance higher than that of adding increment from each of flexibility. Compared with rigid motion, 10% deformation in chordwise and spanwise directions with zero phase angle can enhance the thrust force coefficient from 0.22 to 0.38 and 0.54 respectively, while thrust force coefficient is 0.8 by combination of spanwise and chordwise flexibility.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BF Microfluidics: Devices II 101F

10:30AM BF.00001 Defining Microstructured Fluids and Soft Materials Bottom Up , LIAN LENG, SIAVASH ASLANBEIGI, AXEL GUENTHER, University of Toronto — We demonstrate a microfluidic strategy for the three-dimensional organization of soft bulk materials with a tunable microstructure. Two miscible fluid streams entered a massively-scaled microfluidic device and were distributed through an array of alternating channels. A soft lithography process was adapted to consistently define microchannels with a hydraulic diameter of 150microns in 500micron thin elastomer substrates, followed by bonding of up to ten such layers in the vertical direction. The resulting microfluidic device contained a total of one hundred microchannels. At the device exit, a complex fluid was continuously extruded as a matrix material where the pores contained the second fluid. Upon leaving the chip, the vascularized matrix was solidified and created a perfusable soft material. The material microstructure and its tunability were characterized using microscale computed tomography.

10:43AM BF.00002 Light-enabled digital microfluidics: A technology leading to a programmable lab on a chip , STEVEN T. WERELEY, HAN-SHENG CHUANG, ALOKE KUMAR, Purdue University — We present a self-driven microfluidic device for droplet manipulations based on an open optoelectrowetting (O-OEW) technique. The proposed O-OEW features dynamic droplet maneuverability and great extensibility due to light-induced virtual electrodes and an open configuration. The device comprises coplanar interdigitated electrodes, a photoconductor, and an insulator on a single substrate. The mechanism behind the O-OEW is dependent on the impedance switching between the photoconductor and the insulator. The photoconductor works as a gate for the equivalent circuit. Under illumination the impedance of the photoconductor decreases, prompting an electrowetting effect due to a high voltage drop in the insulator. Without illumination the impedance of the photoconductor increases, shifting the voltage drop back to the photoconductor layer and shutting off the electrowetting. The illumination induces a localized hydrophilic region on an overall hydrophobic surface, causing an imbalance of surface tension forces and the subsequent liquid droplet movement. By selectively illuminating the platform surface, basic droplet operations are implemented, such as translation, merging, and simultaneous multi-droplet control. Immersing the liquid droplets in oil enhances the movements and prevents serious evaporation. For more high-end applications, an addressable light source, such as a DLP projector, can be employed. The integration will enable the realization of a programmable lab on a chip.

10:56AM BF.00003 Beyond Poiseuille: Over-limiting Fluid Flows through Macroscopically Long Carbon Nanochannels[1] , S. SINHA RAY, A.L. YARIN, Dept of Mech. and Industrial Engr., UIC — Nanotubes and nanochannels have tremendous potential in various fields like drug delivery, DNA segregation, capillary electrophoresis etc. Except coelectrospinning all the methods result in nanotubes sufficiently small in diameter (1-100 nm) but not longer than several micron precluding easy manipulation making them almost unsuitable for installing in nanofluidic devices for studying fluid flow characteristics. In this work we developed macroscopically long ($\sim$1 cm) carbon nanochannels and studied flow characteristics in them. Then, we demonstrated that bi-layer flows of liquid and gas can result in an over-limiting regime, where a higher flow rate of liquid can be achieved as compared to the case when the same liquid flows through the same tube subjected to the same pressure drop and occupies the whole bore. This paradoxical result is because the less viscous gas layer can flow much faster than the underlying liquid layer and entrain the latter via a significant shear stress. The present results show that the over-limiting liquid flows through nanotubes, seemingly resembling a deviation from the no-slip condition, in reality are entrained by a rapidly moving gas layer in bi-layer liquid/gas flows. This quasi-slip phenomenon happens in relatively large nanotubes ($\sim$500 nm) where the no-slip condition holds with sufficient accuracy, which can be beneficial in micro- and nanofluidics, nanoreactors and drug delivery systems, which are the current goals of this team.

[1]This work was partially funded by NSF through grant NIRT CBET-060962.

11:09AM BF.00004 Inertial-Microfluidic Hydrodynamic Lens[1] , YOUNG WON KIM, JUNG YUL YOO, Seoul National University — A hydrodynamic lens is the methodology to focus nano- and micro-particles suspended in liquid medium. We designed and tested a single-stage inertial-microfluidic hydrodynamic lens embodied in a microchip for biomedical and environmental applications. We adopted cylindrical micro-orifices with diameters of 100–300 μm, transporting micro-particles in sizes of 1–16 μm. A numerical study is conducted to provide optimum design rules of the lens system. The lens performances are evaluated in terms of Stokes number considering the particle size, the orifice diameter, and the flow Reynolds number. Micro-particle tracking velocimetry (μ-PTV) adopting Nd:YAG lasers, which freeze flowing particles, are applied, and compared with the numerical simulation in terms of the focused beam diameter. The particle focusing method suggested in this work is fairly simple, sheathless, and free from necessity of other external forces.

[1]This work was supported in part by the second stage of the Brain Korea 21 Project in 2009, and by MOEHRD (KRF-0420-20070058).

11:22AM BF.00005 A Piezoelectric Micropumping Based on D31 Mode , YAKUT ALI, CUIFANG KUANG, JAMIL KAHN, GUIREN WANG, Department of Mechanical Engineering, University of South Carolina, Columbia, SC, USA — A micropumping device has been developed, which may find application in different areas such as blood pumping and chemical reagents dosing in bioengineering or as an efficient thermal management solution scheme in space-constrained electronic devices, due to some of their unique properties such as lower noise generation and ease of miniaturization. In this presentation, liquid pumping effect is reported using a simple valveless piezoelectric dynamic pump in D31 mode based on acoustic streaming principle. The actuator tip configuration is found to have a significant effect on the pumping performance. Quantitative results of maximum local velocity are presented for different tip configuration of the same actuator for comparison. In addition, this work also demonstrates the quantitative measurements of the pumping performance such as the flow rate and pressure head generated as a function of different relevant parameters such as applied electrical field, AC frequency and length of the actuator.

11:35AM BF.00006 Performance predictions for valveless impedance pumps in the microscale: wave speed and time response considerations for the Liebau phenomenon[1] , JOHN MEIER, MORTEZA GHARIB, California Institute of Technology — Valveless pumping through periodic excitation of a pliant tube with geometric asymmetry was first noted by Liebau in 1954. Studies by Hickerson and Gharib (J. Fluid Mech. 2006) and Avrahami and Gharib (J. Fluid Mech. 2008) highlight the role of wave dynamics and resonance in valveless impedance pumps that exploit the Liebau phenomenon. While pulse propagation in fluid filled elastic tubes has been studied for centuries, there are fundamental scaling investigations missing from the literature that are necessary to understand impedance pump behavior. The pump has been shown to function down to scales of 250 μm in tubular systems by Rinderknecht et al. (J. Micromech. Microeng. 2005). We have recently shown that the pump also functions in a planar manifestation, fabricated using multilayer soft lithography, with pump thicknesses on the order of 200 μm. In this study we look at wave propagation and time response in both tubular and planar systems and discuss the effects of the scaling parameter λ/L in the pumping element and its affects on performance and fundamental impedance pump behavior.

[1]Graduate student support graciously provided by the NDSEG.

11:48AM BF.00007 Numerical and experimental study of Newtonian and non-Newtonian flow in a spiral viscous pump , GUSTAF MÅRTENSSON, Mydata Automation AB, ANDREAS GUSTAFSSON, Department of Mechanics, KTH — The need to transport small volumes of viscous media is a vital part of microfluidic devices vital to applications in biotechnology, chemistry and electronics. A novel Archimedian viscous micropump was developed in an attempt to achieve precise and accurate delivery of fluid in a robust and industrially viable package. The pump consists of a two-disc system, where one is patterned with a spiral rectangular channel of variable width and the other is smooth and has a rate of rotation Ω in order to pump the fluid. The width of the channel is variable along its length in order to achieve a constant local Reynolds number and avoid recirculation zones along the spiral, which is described $r = a + b\theta^c$, where r is the radius at the spiral centerline and θ is the angle. Numerical and analytical studies of the proposed model will be presented, exhibiting a linear relationship between the flow Q and Ω. Results from experiments with a simplified prototype will also be presented supporting the analytical and numerical studies.

12:01PM BF.00008 Stabilization of ion concentration polarization using a heterogeneous nanoporous junction , PILNAM KIM, Seoul National University, SUNG JAE KIM, JONGYOON HAN, Massachusetts Institute of Technology, KAHP Y. SUH, Seoul National University — We demonstrate a recycled ion – flux through heterogeneous nanoporous junctions, which induce stable ion concentration polarization (ICP) with an electric field. The nanoporous junctions are based on integration of ionic hydrogels whose surfaces are negatively- and positively- charged for cationic selectivity and anionic selectivity, respectively. It is shown that a 'heterojunction' structure with cationic selective hydrogels (CSH) and anionic selective hydrogels (ASH) can be matched up in a way to achieve continuous ion-flux operation for stable concentration gradient or ionic conductance. Furthermore, the combined junctions can be used to accumulate ions on a specific region of the device.

12:14PM BF.00009 Analysis of Magnetohydrodynamic Flow in Microfluidics , YOGENDRA PANTA, WEI LIN, Youngstown State University — Over the last three decades, numerical and experimental fluid dynamic studies have been well documented for optimization of device performance in general fluid dynamics, prediction and analysis of physiological flows, fluid-structure interactions in biological systems, and effectiveness of drug delivery systems in lab on chip devices. Magnetohydrodynamics (MHD) is a proven and a routinely used technology not only in various industries to heat, pump, stir and levitate fluids but also an innovative potential for making remarkable biosensors. Two typical pilot projects to test, analyze and optimize the MHD effects were designed. Microfluidics channels coupled with MHD in various shapes were fabricated from a thin brass sheet sandwiched between two polycarbonate sheets in which two platinum electrodes were patterned on the channel walls. Ionic solution colored with dye was introduced in the channel to visualize the fluid flow with or without the MHD. The induction and driving of fluid motion in the channel was accomplished by placing magnetic field normal to the applied electric field in order to induce Lorentz forces in the fluid contained in the channel. Experimental data and numerical results were obtained in a good agreement. Flow velocities were obtained linearly increasing with the higher magnetic flux densities. Future work will be focused on the development of MHD biosensors for chemical biology applications.

12:27PM BF.00010 Oscillatory Magnetogasdynamic Slip Flow in a Microchannel , RAMESH AGARWAL, Washington University in St. Louis — The problem of pressure driven Magnetogasdynamic (MGD) slip flow with small rarefaction through a long micro-channel is considered. The flow is driven by steady or oscillatory pressure gradient. The study of MGD flows in microchannels is of great interest since they occur in magnetic thin films and other electromagnetic micro-scale devices. In obtaining the micro-fluidic solutions in the presence of a magnetic field, some additional physical, mathematical and numerical issues need to be considered. These issues deal with the scaling laws for micro-scale MHD flows and the relevant parameters such as Mach number, Reynolds number, Hartmann number, magnetic Reynolds number, and Knudsen number. For planar constant area micro-channel, it is possible to obtain the analytical solutions for both steady and oscillatory pressure driven flows. As physically expected, the higher value of the magnetic field (higher Hartmann number) flattens the velocity profile in the channel.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BG Free Surface Flows II 101G

10:30AM BG.00001 Liquid bridge stability and breakup with a receding contact line , KENNETH BREUER, BIAN QIAN, Brown University — We have used experimental and theoretical methods to study the evolution and pinch-off of a liquid bridge fixed at the upper attachment point, but with a free contact line at the lower attachment point. High speed video shows that the contact line motion consists of two stages: a slow retraction at the beginning and a rapid retraction immediately prior to the bridge pinch-off. During the first stage, the evolution is quasi-static, and only a function of the bridge's height and volume. Agreement between experiments and solutions to the Young-Laplace equation is excellent during this phase. At some point, however, the static solution becomes unstable, and the contact line retreats rapidly, pinching off to form a droplet on the substrate. Theoretical and numerical approaches are used to analyse and predict this motion. Excellent agreement with experiments is achieved using a Tanner-like formulation for the dynamic contact angle.

10:43AM BG.00002 Building water bridges in air: Can Taylor and Melcher's leaky dielectric model explain such a striking phenomenon? , ÁLVARO G. MARÍN, DETLEF LOHSE, Physics of Fluids, University of Twente — A liquid bridge forms when applying a high voltage between two glass beakers full of water. Surprisingly, the water bridge defies gravity even when the beakers are separated at distances of around $2cm$. Experimental measurements have been done to characterize its shape and correlations have been found with the electrical current passing through the bridge. A complicated flow pattern within the bridge has been also visualized using high speed recordings and particle velocimetry measurements will show its origin. Although the presented results are preliminary, some mechanisms are proposed and discussed for the stability of the bridge and the source of the flow, revealing an important role of polarization forces on the stability of the water bridge. The system would therefore behave as an extreme example of a *leaky dielectric* which would permit to make use of the famous model developed by Taylor and Melcher.[1]

[1]J. R. Melcher and G. I. Taylor, *Electrohydrodynamics: A review of the role of interfacial shear stresses*, Annual Review of Fluid Mechanics, vol. 1, 111-146 (1969).

10:56AM BG.00003 Hydrostatics, steady flow, and dynamics in helically-supported capillary channels[1] , DAVID THIESSEN, JERRY OELERICH, KELSEY CLINE, LIKUN ZHANG, Washington State University — Helically-supported capillary channels consisting of water-filled, slightly stretched, horizontal springs constitute an ideal system for the study of the hydrostatics and hydrodynamics of capillary channels. This unique geometry allows for sub-micron measurement of the radial position of the meniscus as a function of axial distance down the channel using digital image analysis. Hydrostatic measurements on a 300 micron diameter channel, with a length-to-diameter ratio of 120, give a profile of channel compliance as a function of axial position. Capillary-driven flow in this channel has been demonstrated at flow rates up to 0.8 mL/min (average axial velocity 0.25 m/s). The flow rate limits in these channels are understood in terms of capillary pressure limits. Larger diameter channels (3 mm) connected to a pressure reservoir are being studied for application to microgravity phase separation. In this case, the channel diagnostics are used to examine droplet impact and absorption into the channel as well as for the detection of capillary waves that emanate from the impact point.

[1]Supported by NSF and NASA.

11:09AM BG.00004 Vibration impact on longwave thermocapillary instability in a binary mixture with the Soret effect , IRINA FAYZRAKHMANOVA, Technion - Israel Institute of Technology, Haifa, Israel, SERGEY SHKLYAEV, Perm State University, Perm, Russia, ALEXANDER A. NEPOMNYASHCHY, Technion - Israel Institute of Technology, Haifa, Israel — We study the influence of vibrations on the longwave Marangoni convection in a layer of a binary mixture with the Soret effect. At the lower solid boundary of the layer a fixed heat flux is specified; the upper free boundary is assumed to be nondeformable. In the absence of vibrations, this problem was analyzed in [1]. The vibration frequency is comparable with the eigenfrequency of the oscillatory mode; its amplitude is sufficiently large to provide finite values of the inertia force. A linear stability analysis is performed by means of the Floquet theory. Several limiting cases are treated analytically. In the parameters region, where an oscillatory instability exists in the absence of the vibration [1], a subharmonic mode is critical. With the increase in the vibration amplitude, substantial decrease in the critical Marangoni number is observed. In the parameters region, where a monotonic mode is found in [1], a synchronous mode emerges. In the latter case, either stabilization or destabilization of the system can take place depending on the problem parameters. [1] A. Oron, A.A. Nepomnyashchy, PRE **69**, 016313 (2004). We acknowledge the support by Israel Science Foundation (grant 812/06) and Israeli Ministry of Science (grant 3-5799).

11:22AM BG.00005 Computation of time-periodic solutions of the vortex sheet with surface tension , JON WILKENING, UC Berkeley, DAVID AMBROSE, Drexel University — I will describe a spectrally accurate numerical method for finding non-trivial time-periodic solutions of nonlinear PDE. We minimize a functional (of the initial condition and the period) that is positive unless the solution is periodic, in which case it is zero. We use adjoint methods (originally developed for shape optimization in fluid mechanics) to compute the gradient of this functional with respect to the initial condition. We then minimize the functional using a quasi-Newton gradient descent method (limited memory BFGS). We use our method to compute families of time-periodic solutions of the vortex sheet with surface tension separating two incompressible, irrotational, immiscible, inviscid fluids. As a starting guess, we use analytically determined time-periodic solutions of the linearized problem about a flat interface with constant vortex sheet strength. 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 manifold of symmetric breathing solutions.

11:35AM BG.00006 Nonlinear dynamics of coiling, and mounding in viscoelastic jets[1] , TRUSHANT MAJMUDAR, THOMAS OBER, GARETH MCKINLEY, Massachusetts Institute of Technology — Free surface continuous jets of non-Newtonian fluids, although relevant for many industrial processes like bottle filling, remain poorly understood in terms of fundamental fluid dynamics. Here we present a systematic study of the effect of viscoelasticity on the dynamics of continuous jets of worm-like micellar surfactant solutions of varying viscosities and elasticities, and model yield-stress fluids. We systematically vary the height of the drop and the flow rate in order to study the effects of varying geometric and kinematic parameters. We observe that for fluids with higher elastic relaxation times, folding is the preferred mode. In contrast, for low elasticity fluids we observe complex nonlinear dynamics consisting of coiling, folding, and irregular meandering as the height of the fall increases. Beyond this regime, the jet dynamics smoothly crosses over to exhibit the "leaping shampoo" or the Kaye effect. Upon increasing the flow rate to very high values, the "leaping shampoo" state disappears and is replaced by a pronounced mounding or "heaping". A subsequent increase in the flow rate results in finger-like protrusions to emerge out of the mound and climb up towards the nozzle. This novel transition is currently under investigation and remains a theoretical challenge.

[1]Procter and Gamble

11:48AM BG.00007 Equilibrium shapes of drops emerging slowly on top of vertical tubes , ABEL LOPEZ-VILLA, ABRAHAM MEDINA, ESIME Azcapotzalco, IPN, FRANCISCO HIGUERA, ETSI Aeronauticos, UPM — In this work we are interested in describing the shape of a single drop which emerges on top of a vertical tube just when it reaches the edge of the tube. We assume that the liquid is injected at a very low, constant, volume flow rate Q. We consider, as a first approximation, that the drop shape can be calculated by using a balance of force equation that relates the hydrostatic and capillary presures. The results are valid for cases of good and poor wetting, respectively, and for high and low Bond numbers. We present experiments that validate the numerical results obtained for the drop shapes.

12:01PM BG.00008 Cowboy Fluid Mechanics: Lariat Modes of a Viscous Rope , NEIL RIBE, Laboratoire FAST, Orsay, France, SARAH BADR, Universite de Paris-Sud, Orsay, France, STEPHEN MORRIS, University of Toronto, Toronto, Canada — A thin filament of viscous fluid falling onto a surface winds itself into a helical coil whose angular frequency of rotation Ω depends on the fall height H, the flow rate, and the fluid properties. We have studied a novel variant of this phenomenon in which the nozzle ejecting the fluid rotates about a vertical axis at a constant rate ω. In laboratory experiments using viscous corn syrup, we observe that the filament coils in the normal way when $\omega \ll \Omega$. However, when $\omega \approx \Omega$ and H is sufficiently large, a new "lariat" mode appears in which the filament is thrown outward in the form of a spiral of large diameter (up to tens of cm) rotating at a rate $\approx 0.9\omega$. The transition between the coiling and lariat modes is hysteretic with respect to variations in ω. In addition to the laboratory experiments, we will also present preliminary results of numerical calculations of the lariat mode based on a "slender body" model for a viscous filament with inertia.

12:14PM BG.00009 An other viscous catenary , JACOPO SEIWERT, MARIE LE MERRER, DAVID QUÉRÉ, CHRISTOPHE CLANET — The classical viscous catenary is obtained by the gravitational deformation of an initially horizontal thread of viscous fluid (honey for example). A second shape exists, roughly composed of three perpendicular pieces (like a flying trapeze). We study the physics of this flying trapeze-shape and show in particular that its origin is independent of the liquid viscosity, and is only fixed by the geometric characteristics (length and radius) of the (initial) filament.

12:27PM BG.00010 ABSTRACT WITHDRAWN –

Sunday, November 22, 2009 10:30AM - 12:40PM –

Session BH Drops II 101H

10:30AM BH.00001 Role of Gas Composition in Viscous Drop Splashing , CACEY S. STEVENS, SIDNEY R. NAGEL, James Franck Institute, University of Chicago — Splashing occurs when a liquid drop impacts on a smooth, dry surface at high velocity. It has been discovered that the pressure of the surrounding air is important in causing a splash.[1] Moreover, viscous splashing occurs at a much later time than the splashing of an inviscid liquid.[2] Here we investigate the relationship between the threshold pressure, P_T, and the impact velocity, V_0, for viscous splashes. P_T demonstrates only a weak dependence on impact velocity above 2.0 m/s. Also, the composition of the gas in which splashing occurs is varied to gain insight on the role of gas molecular weight on P_T in the viscous regime.

[1]L. Xu, S. Nagel, and W. Zhang. Phys. Rev. Lett. 94, 184505 (2005).
[2] L. Xu, Phys. Rev. E 75, 056316 (2007).

10:43AM BH.00002 Drop Impact of Viscous Suspensions on Solid Surfaces , DANIEL BOLLEDDULA, ALBERTO ALISEDA, Department of Mechanical Engineering, University of Washington — Droplet impact is a well studied subject with over a century of progress. Most studies are motivated by applications such as inkjet printing, agriculture spraying, or printed circuit boards. Pharmaceutically relevant fluids provide an experimental set that has received little attention. Medicinal tablets are coated by the impaction of micron sized droplets of aqueous suspensions and subsequently dried for various purposes such as brand recognition, mask unpleasant taste, or functionality. We will present a systematic study of micron sized drop impact of Newtonian and Non-Newtonian fluids used in pharmaceutical coating processes. In our experiments we extend the range of Ohnesorge numbers, $O(1)$, of previous studies on surfaces of varying wettability and roughness.

10:56AM BH.00003 Effect of parameters on redistribution and removal of particles from drop surfaces , SAI NUDURUPATI, MUHAMMAD JANJUA, LSSU, PUSHPENDRA SINGH, NJIT, NADINE AUBRY, CMU — It was recently shown by us that particles distributed on the surface of a drop can be concentrated at the poles or the equator of the drop by subjecting it to a uniform electric field and that such concentrated particles can then be removed from the drop by increasing the electric field intensity. In this talk, we present experimental results for the dependence of the dielectrophoretic force on the parameters of the system such as the particles' and drop's radii and the dielectric properties of the fluids and particles, and define a dimensionless parameter regime in which the technique works. In particular, we show that the technique is guaranteed to work if the drop radius is smaller than a critical value that depends on the physical properties of the drop and ambient fluids and those of the particles.

11:09AM BH.00004 Particle deposition during evaporation of colloidal sessile drops , JAMES D. FELSKE, State University of New York at Buffalo, HASSAN MASOUD, Georgia Institute of Technology — Deposition patterns of particles suspended in evaporating colloidal drops are determined by the flow fields within the drops. Using analytically determined velocities, particle motions are then tracked in a Lagrangian sense. It is found that the majority of particles intersect the free surface as it recedes. Such "capture" of particles by the free surface is found to be the major mechanism in establishing the deposition pattern. Patterns are calculated for wetting and non-wetting drops whose contact lines are either pinned or freely moving during evaporation. The distribution of evaporative flux which drives the flows is taken to be that engendered by gas-phase diffusion. The theoretical results are found to agree favorably with available experimental data.

11:22AM BH.00005 Generation of Single, Monodisperse Compound Droplets[1] , JAMES BLACK, PRASAD BHAVE, Georgia Institute of Technology, OUMAR TOURÉ, Polytech Clermont-Ferrand, 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 thermocapillarity. 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. An apparatus was designed to produce single compound droplets of variable water/oil volume ratio and uniform size. Experiments were performed to characterize the effect of apparatus and fluid parameters on the volume ratio in the generation of droplets of silicone-oil-encapsulated water.

[1]Supported by NASA and NSF

11:35AM BH.00006 The mayonnaise droplet[1] , DENIS TERWAGNE, TRISTAN GILET, NICOLAS VANDEWALLE, STÉPHANE DORBOLO, University of Liège, GRASP-PHOTOPÔLE/PHYSICS DEPARTMENT TEAM — A compound drop is made of a millimetric water drop encapsulated by an oil shell. They are obtained by merging one drop of each component (water and oil). Afterwards, they are laid on a high viscosity oil bath which is vertically vibrated. When the forcing acceleration is higher than a given threshold, compound drops can bounce on the surface. We show that above a second threshold some oil contained in the shell enters in the inner water droplet. In a second experiment, we drop the compound droplet on the oil bath at rest. We can determine the range of impact speed in which capillary waves developed on the surface are able to generate an oil drop (coming from the shell) in the water drop. When the bouncing trajectories of the droplets are analyzed a correlation between the emulsion threshold and the static analysis can be made.

[1]Authors thank the FNRS/FRIA for financial support. Part of this work has been supported by COST P21 "Physics of droplets" (ESF).

11:48AM BH.00007 Impact and Spreading of a Compound Droplet: A Model for Single Cell Epitaxi , SAVAS TASOGLU, UC, Berkeley Ph.D. Student, GOZDE KAYNAK, Koc University, M.S. Student, METIN MURADOGLU, Koc University, Associate Professor — In recent years, there has been a growing interest in generating compound droplets mainly due to their potential commercial value [1] and applications in emerging technologies such as single cell epitaxi [2]. Ejecting encapsulated cells on a rigid surface is a promising way to produce 2D/3D tissues [2]. However, this gained experimental capability requires a true understanding of the impact dynamics of the encapsulated cells on solid surfaces for further development. In the present study, a finite-volume/front-tracking method is used to model the impact and spreading of a viscous compound droplet on a flat solid surface as a first step in developing a model for the single cell epitaxi. The cell, the encapsulating droplet and ambient fluid are all assumed to be Newtonian. Simulations are performed for a range of dimensionless parameters and their effects on deformation of inner droplet are investigated. These results provide initial insight about the optimum parameter ranges for highest viability of cells. [1] Utada, Lorenceau, Link, et al., Science, 308(5721), (2005). [2] Demirci and Montesano, Lab Chip, 7, (2007).

12:01PM BH.00008 Dynamics of the electro-osmotically toggled droplet switch , DOMINIK BARZ, Forschungszentrum Karlsruhe GmbH, IKET, D-76021 Karlsruhe, Germany, MICHAEL VOGEL, PAUL STEEN, Cornell University, School of Chemical & Biomolecular Engineering, Ithaca NY 14853, USA, EODS COLLABORATION — A capillary switch is a bi-stable system of liquid/gas interfaces with a trigger to toggle back and forth between the energy wells. The electro-osmotic droplet switch consists of a pair of coupled droplets whose shape-change is triggered by volume transfer using an electro-osmotic pump placed between the droplets. With the pump off, the stable states are a pair of identical sub-hemispherical drops for low total-volume or a large-small droplet configuration(two mirror-symmetric states) for large enough total-volume. With the pump on, these stationary states are shifted or there are no stationary states at all, depending on the pump strength. We report the pump-on behavior as a modification of the pump-off behavior. A dynamic model of the system which adopts an Ergun-equation characterization of the pump is introduced. Model predictions are compared to observation.

12:14PM BH.00009 Digital microfluidics by using Electric Charging of Droplet[1]

, YONG-MI JUNG, IN SEOK KANG, POSTECH — We evaluate the novel actuation of droplets by exploiting Coulombic forces acting on conductive droplets immersed in a dielectric fluid. The droplet under a DC voltage can be charged by direct contact, by charge transfer along an electrical conducting path, or by both mechanisms. A droplet is deformed to a shape with a sharper tip near electrode. The droplet acquires charges through it and may then be transported rapidly by exploiting Coulombic forces. For actuation by this phenomenon, we construct a platform for handling droplets on the device by using electrode dots arrayed on substrate. A programmable voltage supply is connected through an electrical switch to the electrodes. In the experimental results, droplets are actuated along the electric field signal. The droplet never adhere surface of chip when it moves. We also perform electrocoalescence and chemical reaction in a merged droplet as an essential step for being used as droplet-based microreactors. When two droplets approach each other due to Coulombic force, a liquid bridge is formed between them. The chemical reaction simultaneously occurs by coalescence of droplets.

[1]This work was supported by the grant R01-2008-000-20848-0 from KOSEF, by a grant from CUPS sponsored by KOSEF, and by the BK21 program of the Ministry of Education of Korea.

12:27PM BH.00010 Bouncing of a Droplet on Superhydrophobic Surface in AC Electrowetting[1]

, KWAN HYOUNG KANG, SEUNG JUN LEE, JIWOO HONG, Pohang University of Science and Technology — Oscillation of a droplet is induced in ac electrowetting by time-dependent electrical wetting tension. A droplet placed on a superhydrophobic surface bounces up like a rubber ball when an ac signal is applied. The bouncing pattern is highly frequency dependent. We investigated how the shape deformation and bouncing of a droplet are affected by applied frequency. The droplet motion is analyzed with the spectral method. The droplet is modeled as a simple linear oscillator, and the mass and spring constants are determined based on analytical results. We found that bouncing occurs periodically at a resonance frequency of the droplet. The motion of a sessile droplet released from a fixed shape is analyzed based on the phase field method. The numerical results show qualitative agreement with the experimental results for a bouncing droplet. Details on the flow field inside a bouncing droplet will be discussed based on numerical results.

[1]This work was supported by the Korea Research Foundation Grant funded by the Korean Government (MOEHRD, Basic Research Promotion Fund) (KRF-2006-331-D00058).

Sunday, November 22, 2009 10:30AM - 12:40PM –

Session BJ Bubbles II: Bubble Deformation and Breakup 101I

10:30AM BJ.00001 Viscous irrotational analysis of the deformation and break-up time of a bubble or drop in uniaxial straining flow[1]

, JUAN C. PADRINO, DANIEL D. JOSEPH, University of Minnesota — The deformation of a bubble or drop in uniaxial straining flow is modeled by assuming the motion of the viscous fluids to be irrotational. In this approach, viscosity enters the analysis through the balance of normal stresses at the interface. The tracking of the interface motion is achieved by integration of the set of differential equations furnished by the conservation of linear momentum and kinematical conditions, coupled with a boundary-integral formulation along the interface. This methodology is thus employed to investigate the influence of a finite Reynolds number on the time evolution, up to break-up, of the drop or bubble for various Weber numbers. Comparison with results from simulations of the Navier-Stokes motion, which includes rotational effects, are presented and discussed.

[1]Acknowledge support from NSF - Computational resources provided by the Minnesota Supercomputer Institute for Advanced Computational Research.

10:43AM BJ.00002 Bubble deformation, translation, collapse and bursting in a uniform electric field[1]

, STEPHEN SHAW, PETER SPELT, OMAR MATAR, Imperial College London — We study the dynamics of a bubble in a dielectric fluid subjected to a uniform electric field in the limit of weak viscosity and compressibility. We use a domain perturbation method to derive a set of ordinary differential equations that govern the evolution of the bubble deformation, oscillation and translation; these equations contain second-order interaction terms. Both steady-state and time-dependent solutions are presented. Our analysis of the results indicates that for initially uncharged spherical bubbles, only even shape modes and odd components of the charge density are excited. We show that situations wherein all modes are excited could give rise to an instability over a certain range of parameter values. This instability manifests itself via suddent bubble acceleration and growth, which ultimately renders the theory invalid.

[1]EPSRC Grant number EP/D50371X/1

10:56AM BJ.00003 Bubble pinch-off in viscous liquids[1]

, ROCÍO BOLAÑOS-JIMÉNEZ, ALEJANDRO SEVILLA, CARLOS MARTÍNEZ-BAZÁN, Universidad de Jaen, DEVARAJ VAN DER MEER, University of Twente, JOSÉ MANUEL GORDILLO, Universidad de Sevilla — The effect of liquid viscosity on the final instants previous to pinch-off of an air bubble immersed in a stagnant viscous liquid is experimentally and theoretically investigated. Our experiments show that the use of a power-law to describe the collapse dynamics of the bubble is not appropriate in an intermediate range of liquid viscosities, for which a transition from an inviscid to a fully viscous pinch-off takes place. Instead, the instantaneous exponent $\alpha(\tau)$ varies during a single pinch-off event from the typical values of inviscid collapse, $\alpha \simeq 0.58$, to the value corresponding to a fully viscous dynamics, $\alpha \simeq 1$. However, we show that the pinch-off process can be accurately described by the use of a pair of Rayleigh-like differential equations for the time evolution of the minimum radius and the axial curvature evaluated at the minimum radius, r_1. This theoretical model is able to describe the smooth transition which takes place from inviscid to viscous-dominated pinch-off in liquids of intermediate viscosity, $10 \leq \mu \leq 100$ cP, and accounts for the fact that the axial curvature remains constant when the local Reynolds number becomes small enough, in agreement with our experimental measurements.

[1]Supported by the Spanish MCyI under project DPI2008-06624-C03-02.

11:09AM BJ.00004 Evolution and Pinch-off of Axisymmetric Viscous Bubbles in Stokes Flow

, SHADI NADERI, MONIKA NITSCHE, University of New Mexico — Boundary integral simulations of the evolution of an axisymmetric viscous bubble in an axisymmetric strain field are presented, using the Stokes flow approximation. Previous works have shown the bubble reaches a steady state while the capillary number measuring the strain field stays below a critical value. Above this value no steady state is found. Previous experimental observations and numerical works indicate that a slight increase in the capillary number past this critical value causes an elongation and break-up. We present numerical studies of the evolution of bubbles towards the steady state subject to various capillary numbers and viscosity ratios using the high order method developed by Ceniceros, Karniala and Nitsche (preprint). Steady state results obtained from this method are compared with previous findings. A numerical investigation of pinch-off for a capillary number past this critical value is presented.

11:22AM BJ.00005 X-ray Imaging of Memory in Air Bubble Pinch-Off , NATHAN KEIM, James Franck Institute, U. of Chicago, KAMEL FEZZAA, Advanced Photon Source, Argonne Nat'l Laboratory, SIDNEY NAGEL, James Franck Institute, U. of Chicago — We report on studies of underwater air bubble pinch-off. Previously, we have shown that this pinch-off is a singularity with memory, in which azimuthal symmetry may be broken by tilting the nozzle or by blowing bubbles from a slot-shaped nozzle, and that 2- or 3-lobed perturbations to the pinching neck's cross-section are remembered as small vibrations of the neck shape.[1] This is consistent with the model of Schmidt et al.[2] Even modest perturbations to the initial bubble shape can cause the neck to develop concavities late in its collapse, as shown by Turitsyn et al.[3] Using high-speed X-ray phase contrast imaging at the Advanced Photon Source, we have observed these concavities, as well as the Worthington jet. Tilting a nozzle by as little as 1° suppresses jet formation outside of a small region near pinch-off. Further experiments show that placing walls near the bubble also creates azimuthal perturbations, and that the vertical motion and vertical asymmetry of the neck at its minimum radius are due primarily to the neck's impedance of gas flow.

[1]Keim & Nagel, DFD 2008 AH.1
[2]*Nat. Phys.***5**, 343
[3]arXiv:0902.0393v1

11:35AM BJ.00006 Diffraction: wave dynamics near the break-up of an underwater bubble , LIPENG LAI, SAMUEL D. OBERDICK, WENDY W. ZHANG, University of Chicago — Recent studies show that the final form of bubble break-up is dominated by a memory of initial asymmetries, in contrast to the idea that the break-up dynamics inevitably evolves towards a universal form independent of boundary and initial conditions. Specifically, when the neck of the submerged bubble is distorted from cylindrical symmetry, vibrations in the neck cross-section about the average contraction are excited. These vibrations, which correspond to standing waves on the air-water interface, persist until break-up. As a result, the break-up dynamics is both asymmetric and dependent on the initial distortion. Previous works analyzed the situation where the initial distortion is dominated by a single vibrational mode. They found that the final dynamics continues to be dominated by the initial mode. Nonlinear interactions, which create new modes and change the relative amplitudes of the different modes present, are expected to have a more pronounced role in the diffraction limit, when many vibrational modes are present. Here we examine the break-up dynamics when a large number of vibrational modes are present. We use a boundary integral simulation to track the how the cross-section of the bubble neck evolves over time. The results are compared against predictions from linear stability.

11:48AM BJ.00007 Effect of vertical flow on the break-up of an underwater bubble , DANIEL C. HERBST, WENDY W. ZHANG, University of Chicago — Previously, the break-up of a fluid drop was believed to evolve towards a universal singularity, with little dependence on initial or boundary conditions. Recent studies reveal that the break-up of a bubble while it is immersed in another liquid follows a different scenario, one preserving detailed information about the initial state. When the bubble neck is nearly cylindrical, the leading-order dynamics has a simple two-dimensional form: the initial shape is advected inwards by a focusing flow in the exterior until break-up. Asymptotic analysis indicates that such a memory-preserving evolution is possible only when the vertical flow out of the minimum remains far weaker than the focusing flow within a horizontal plane. Here we explore what happens to this memory-preserving dynamics when the vertical flow becomes comparable with the horizontal flow. An axisymmetric boundary integral code is used to track the shape evolution. We alter the surface profile at break-up, in particular the upper and lower cone angles, by changing the initial neck shape. For large angles, the vertical momentum flux becomes significant and the velocity evolution is strongly coupled to the surface evolution. We also study the effect of an up-down asymmetry in the initial shape on the final break-up.

12:01PM BJ.00008 Combinations of neck vibrations in bubble break-up , SAMUEL D. OBERDICK, LIPENG LAI, University of Chicago, KONSTANTIN S. TURITSYN, Los Alamos National Lab, WENDY W. ZHANG, University of Chicago — When an air bubble pinches off inside a liquid, the final dynamics is controlled by the initial shape asymmetries, via vibrations in the neck cross-section. Previously, we showed that, when the break-up is dominated by 1 vibrational mode, the initial shape asymmetry evolves into a smooth contact that divides the cross-section shape into side lobes. The lines of symmetry remain static over time and the average size of the side lobes decreases in discrete steps as the initial distortion size is reduced. Here we use analytics and simulation to study the contact dynamics obtained by combining 2 vibrational modes. A wide variety of intricate contact shapes are possible. When incommensurate modes are present, the lines of symmetry are destroyed. The contact shape becomes askew, i.e. the surface on opposite sides of the contact have different curvature values. When the modes are commensurate, some symmetry lines are preserved. Interference between the different vibration frequencies causes the surface distortion to vary irregularly over time. As a result, the average size of the side lobes decreases in irregular steps as the initial distortion size is reduced.

12:14PM BJ.00009 Non-axisymmetric collapse of cylindrical cavities , IVO PETERS, OSCAR R. ENRIQUEZ PAZ Y PUENTE, STEPHAN GEKLE, LAURA E. SCHMIDT, DEVARAJ VAN DER MEER, DETLEF LOHSE, University of Twente, The Netherlands — Upon the impact of a circular disk on a water surface an expanding cylindrical cavity is created which collapses under the influence of the hydrostatic pressure. We experimentally observe small disturbances in the azimuthal direction that tend to grow towards the pinch-off. To quantitatively investigate the growth of specific mode-numbers, we use disks with a harmonic disturbance applied to their round shapes and study the collapse of the disturbed cavity using high-speed imaging. We performed experiments using disturbances up to mode number $m = 6$, with varying strength from 1% to 25% of the radius of the undisturbed circular disk. For the smallest disturbances we compare the experimental results to a linear stability analysis, following Schmidt *et al.*, Nat. Phys. 5, 343-346 (2009). Larger disturbances become non-linear in an early stage, showing a wealth of complex phenomena like secondary collapses and jets, during which the initial symmetry corresponding to the mode number m always remains preserved.

12:27PM BJ.00010 Collapse of cylindrical vapor cavities in a compressible fluid , DERRICK TREICHLER, KEN KIGER, University of Maryland — The collapse of infinitely long cylindrical vapor cavities in water is studied computationally using Gemini, a compressible hydrocode developed by NSWC/IHD. Simulation results are compared to dynamics given by the cylindrical analogue of the Rayleigh-Plesset equation for spherical bubble dynamics. The results of the incompressible solution are known to depend on size of the domain due to a logarithmic dependance in the governing equation. Compressibility is shown to be a controlling factor in the dynamics of the cavity collapse, both as a means to limit the amount of fluid mass to be accelerated and as a source of radiated energy. As a result, the compressible case reaches an invariant collapse time for fluid domains large enough that acoustic waves traveling outward from the cavity wall are unable to return to the bubble before collapse. Analytical results predict a monotonically increasing collapse time with increasing fluid domain size. Thus, for sufficiently large fluid domains, the analytical solution greatly over-predicts the cavity collapse time given by the computational results.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BK Multiphase Flows II 101J

10:30AM BK.00001 The Impact of Density Ratio on the Primary Atomization of a Turbulent Liquid Jet in Crossflow[1] , 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. Although correlations derived from experimental data exist for the jet penetration, predicting the drop size distribution resulting from the primary breakup of the liquid jet is a more challenging task. Furthermore, most correlations are derived from experimental data performed at ambient conditions, thereby not matching the density ratio found in most gas turbine applications. In this paper, we will study the impact of density ratio on the primary atomization of a turbulent liquid jet injected into a subsonic crossflow using detailed numerical simulations, leaving constant all other relevant characteristic numbers, i.e. jet and crossflow Weber numbers, Reynolds numbers, and momentum flux ratio. The influence of density ratio on the physical mechanisms causing the initial breakup of the jet, the resulting grid dependent/independent drop size distributions, and the jet penetration will be discussed.

[1]This work was supported in part by Cascade Technologies Inc. under Navy SBIR N68335-07-C-0282.

10:43AM BK.00002 Modeling Electrohydrodynamic Atomization[1] , BRET VAN POPPEL, OLIVIER DESJARDINS, JOHN DAILY, University of Colorado at Boulder — Over the past decade, there has been considerable interest in controlling the emissions from small engines in the size range of 200 cm^3 or smaller. Fuel injection schemes may reduce the incidence of pollutant emissions within this class of engines. However, the cost of implementation is a barrier to large scale adoption. One approach to small-scale fuel injection is to capitalize upon the benefits of electrohydrodynamics (EHD) and enhance fuel atomization. There are many possible benefits to EHD aided atomization for combustion, such as smaller droplets, wider spray cone, and the ability to control or "tune" the spray for improved performance. In this work, we perform detailed numerical simulations of EHD-aided liquid breakup in the context of Diesel injection using the new adaptive spectrally refined interface (ASRI) tracking method coupled to a robust and accurate Navier-Stokes/Ghost fluid solver. This novel interface tracking methodology provides several features that improve the accuracy and resolution of the liquid structures. Relevant parameters, such as Weber number, electric Reynolds number, electric Bond number, and charge ratio, are varied to assess the effect of EHD on primary atomization.

[1]The authors gratefully acknowledge financial support from Army Research Office.

10:56AM BK.00003 Detailed numerical simulation of primary atomization processes of liquid fuel jet , JUNJI SHINJO, Japan Aerospace Exploration Agency, AKIRA UMEMURA, Nagoya University — In order to elucidate the physical mechanisms of primary atomization of liquid fuel jet, very detailed numerical simulations have been conducted that resolve all the relevant scales in the primary atomization regime. Three cases of Diesel- like fuel injection into quiescent air with different injection velocities are compared to see the breakup mechanism and the effect of Weber and Reynolds numbers. Surface instability development, ligament creation and droplet formation are observed in the results and characterization of each process has been done. As the injection velocity is increased, the length scale of liquid structure becomes smaller. Ligament formation is observed in the regime of O(1) local Weber number. Droplet formation is governed by the dynamic effect of surface tension waves as expected by our previous study using slow laminar liquid jets. Extension to modeling using these findings will be also discussed.

11:09AM BK.00004 Windswept droplets , KEVIN NJIFENJU, JOSE BICO, PMMH-ESPCI, Paris, EMMANUELLE ANDRES, Technocentre Renault, Guyancourt, MARC FERMIGIER, PMMH-ESPCI, Paris — A droplet deposited on a flat surface (e.g. a car windshield) tends to remain pinned due to contact angle hysteresis. However the droplet may be displaced when sheared by airflow or under the action of gravity. We investigate the conditions that enable the motion of a windswept droplet and describe the liquid dynamics as a function of the combined action of wind shear, gravity and capillary forces. In particular we show that these dynamics can be rationalized in terms of non-dimensional Weber, Bond and Capillary numbers. We finally describe the interaction between multiple moving droplets.

11:22AM BK.00005 Evolution of a polydispersed spray in heated and in highly turbulent flow , FLORIAN MOREAU, RUDY BAZILE, Institut de Mecanique des Fluides de Toulouse - France — This work aims to study experimentally the dispersion and the evaporation of a polydispersed and bi-component spray in highly turbulent and heated flow. A chamber is designed to generate a heated turbulent flow in which two-component droplets are injected. The two components are octane (85%) and 3-pentanone (15%) and are chosen such that the 3-pentanone vapour concentration can be characterized by laser techniques. The experimental setup consists of a vertical channel with optical access. Before the heated air is injected in the channel, it passes through a turbulence generator. The carrier flow is characterized using Laser Doppler Anemometry. The turbulence is shown to have isotropic properties after a distance equal to four times the width of the channel and to have high levels up to 30%. The liquid phase is characterized with Phase Doppler Anemometry which allows to measure the diameter, the longitudinal and the radial velocity of the droplets. The spatial evolution of the diameter probability density function (PDF) and of the rms and mean velocities are obtained. Droplets mass fluxes are also calculated. In the mixture, 3-pentanone is the only component that fluoresces. So the vapour concentration of 3-pentanone in the carrier flow is determined using Laser Induced Fluorescence.

11:35AM BK.00006 Lattice Boltzmann Simulations of a Falling Droplet on a Rest Fluid Film[1] , YUEHONG QIAN, Shanghai University, KE ZHANG, XUESHENG CHU, KAI YAN, Ship Scientific Research Center of China, Wuxi, 214082 — A single-phase model based on lattice Boltzmann [1,2] method is used to investigate the motion of the free surface. To describe the topological deformation of the fluid interface, the cell in the single-phase free surface model is divided into three types: fluid cells, interface cells and the empty cells. The distinctive feature of the model is that the propagation and interaction processes are carried out only in the interface cell and the fluid cell. Numerical simulations of a droplet falling onto a resting fluid film [3] is presented. The Crown formation shown in figure 1 as well as the splashing droplets have been found at different dimensionless Reynolds and Weber numbers, Some comparison with experiment will be also made. REFERENCES [1] Y.H. Qian, D. D'Humières, P. Lallemand. Lattice BGK models for Navier-Stokes equation. Europhys. Lett 1992(17): 479-484. [2] N. Thurey, U. Rude. Interactive free surface fluids with the lattice Boltzmann method. Technical report 2005. University of Erlangen-Nuremberg, Germany. [3] Z.Y. Shi, Y.H. Yan, F. Yang, Y.H. Qian and G.H. Hu. A lattice Boltzmann method for simulation of a three dimensional drop impact on a liquid film. Journal of Hydrodynamics 2008,20 (3):267-272.

[1]Supported by IRT0844, 10625210, 08ZZ43, 07PJ14041.

11:48AM BK.00007 Modeling Interaction between Impinging Droplet and Wall with Lattice Boltzmann Method , YONG LI, TIAN TIAN, Massachusetts Institute of Technology — The fluid mechanics of droplet wall interaction is of great importance in many applications. In different parameter range, the results could be totally different depends on the different magnitude of influence among surface tension, inertial force and viscous force. Experiment observation shows many kinds of interactions such as splashing, sticking, rebounding. The Lattice Boltzmann Method has shown promising progress in simulation multiphase flow. But the small density variation requirement limits the parameter range it can model. An enhanced Lattice Boltzmann Method is introduced to extend the applications to large droplet air density ratio ($\sim$1000) and high velocity ($\sim$30m/s) range. The parameter zones associated different phenomenon zones are explored. Modeling results explain the forming mechanisms of phenomena. Using droplet interaction with small feature on solid surface, the model capability is demonstrated.

12:01PM BK.00008 Collision between immiscible drops with large surface tension difference , MARCO ARIENTI, XIAOYI LI, MARIOS SOTERIOU, United Technologies Research Center, MARK SUSSMAN, Florida State University — Immiscible drop collision, as occurring in fuel-oxidizer sprays or in the release of certain fire-extinguishing agents, tends to exhibit a much richer behavior with respect to miscible drops collision thanks to the formation of a liquid-liquid interface during impact. For instance, in near-head-on diesel-water drop collisions, "overlaying" may occur in which the diesel oil flows from the collision point around the water drop to gather at the opposite location of the drop. To simulate this class of multi-material flows, the combined volume-of-fluid / level set methodology that sharply captures a single liquid-gas interface (*Sussman et al, J. of Comp. Phys., 2007*) needs to be duplicated for a second, independent interface. In this presentation, we will show that simulation results are not affected by the reconstruction order of the interfaces, as in other surface capturing methods. We will also propose different numerical solutions to treat surface tension in the triple point computational cells, and examine the characteristics of the flow developing at the contact line between the two liquids and air in overlaying head-on collisions.

12:14PM BK.00009 Study of Mass Distribution from Two Phase Unlike Impinging Injectors , RAKESH PRABHAKARAN, B.N. RAGHUNANDAN, SOWMYA BOLAKONDA, I.I.Sc — Two phase impinging injectors as an alternative to conventional coaxial injectors in propulsive devices offer many advantages. In addition to simplicity of design and fabrication, spray shaping according to the need is possible with gas-liquid impingement. The fact that mass distribution can be varied as desired is the main theme of this study with air and water as working fluids. In the doublet configuration, the condition of the gas jet is varied and its effect on the mass distribution is studied. As can be visualized, the circularly symmetric spray mass distribution gets distorted in the presence of the gas jet. Even at low pressure ratios, near elliptical mass distribution results. As gas pressure increases, there is a tendency for the mass distribution to be shifted in the direction of gas jet. The effect of some of the geometric parameters on the mass distribution as well as drop-size distribution are studied. Mechanistic details of jet penetration and the inherent instability in the impinging system are discussed. The data base generated is expected to help designers in spray shaping applications.

12:27PM BK.00010 Injection and Disruption of Supersonic Droplets[1] , Y.J. KIM, R.G. CERFF, J.C. HERMANSON, University of Washington — The disruption of simulated fuel droplets in supersonic flow is examined experimentally in a draw-down supersonic wind tunnel. Mono-disperse 100 μm diameter neat fluid droplets are generated using a droplet-on-demand generator upstream of the tunnel entrance. The droplets are accelerated in the supersonic flow, achieving supersonic velocities relative to the surrounding air. The droplets are imaged by direct close-up single- and multiple-exposure imaging. The latter technique allows measurement of the droplet velocity, from which the Mach number relative to the droplet, as well as the Weber number, are determined. The droplets reach a relative Mach number of as high as 1.7 and Weber numbers as high as 260. Droplet deformation and breakup patterns for these conditions can be classified into four different flow regimes by considering the changes in the Weber with downstream distance as the droplet accelerates. The drag coefficients associated with the droplet disruption under locally supersonic conditions are generally higher than those expected for solid spheres.

[1]Supported by the National Science Foundation.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BL CFD II: Methods 200A

10:30AM BL.00001 A finite-volume contact-capturing scheme , ARPIT TIWARI, RATNESH SHUKLA, CARLOS PANTANO, JONATHAN FREUND, University of Illinois at Urbana-Champaign — Finite-volume schemes have been remarkably successful at capturing shock waves. Here, the local characteristics, of course, move into the shock, so in a sense these shock capturing schemes have the task of adding sufficient dissipation so that the captured shock remains relatively sharp but also sufficiently resolved to be compatible with the underlying discretization. In a similar spirit, we have developed a scheme for capturing contact discontinuities, with our particular interest being phase or material boundaries. These are fundamentally different than shocks because the characteristics are parallel to the contact, so a capturing scheme needs to counter the numerical diffusion (as needed for shocks) that thickens the contact in time. In analogy to the well-placed dissipation in shock capturing, terms are added to the equations that also capture these contacts over long times as sharp near-jumps, similar to captured shocks. These terms preserve both sharp mass and material jumps and are compatible with various equations of state, including interfaces between perfect gas and Mie-Grüneison materials. The algorithm is demonstrated in various bubble and pore collapse scenarios.

10:43AM BL.00002 Analysis of Explicit Algorithms for Fluctuating Hydrodynamics[1] , A.C. DONEV, LBNL, E. VANDEN-EIJNDEN, Courant Institute, NYU, A. GARCIA, Dept. Physics & Astronomy, SJSU, J. BELL, LBNL — We describe the development and analysis of finite-volume methods for the Landau-Lifshitz Navier-Stokes (LLNS) equations and related stochastic partial differential equations in fluid dynamics. The LLNS equations incorporate thermal fluctuations into macroscopic hydrodynamics by the addition of white-noise fluxes whose magnitudes are set by a fluctuation-dissipation relation. We introduce a systematic approach based on studying discrete equilibrium structure factors for a broad class of explicit linear finite-volume schemes. This new approach provides a better characterization of the accuracy of a spatio-temporal discretization as a function of wavenumber and frequency, allowing us to distinguish between behavior at long wavelengths, where accuracy is a prime concern, and short wavelengths, where stability concerns are of greater importance. We use this analysis to develop a specialized third-order Runge Kutta scheme that minimizes the temporal integration error in the discrete structure factor at long wavelengths for the one-dimensional linearized LLNS equations. Together with a novel random-direction method for evaluating the stochastic fluxes in dimension larger than one, our improved temporal integrator yields a scheme for the three-dimensional equations that satisfies a discrete fluctuation-dissipation balance for small time steps and is also sufficiently accurate even for time steps close to the stability limit.

[1]A. Donev's work was performed in part under the auspices of the U.S. DoE by LLNL under Contract DE-AC52-07NA27344.

10:56AM BL.00003 High-Order Finite-Difference Solution of the Poisson Equation with Interface Jump Conditions , ALEXANDRE MARQUES, JEAN-CHRISTOPHE NAVE, RODOLFO ROSALES, Massachusetts Institute of Technology — The Poisson equation with jumps in function value and normal derivative across an interface is of central importance in the numerical study of multi-phase flows. In this presentation we introduce a method to obtain a high-order solution to such problem. The method is based on the construction of corrector functions that provide accurate extensions of the jump conditions around the interface. The accuracy of the method results from the combination of Hermite interpolants and a high-order representation of the interface using the gradient-augmented level-set technique. These corrector functions can be easily incorporated in standard finite-difference discretization schemes, only generating additional terms to the right-hand side of the system. As a result, computational cost is not significantly affected when compared to the first order accurate ghost fluid method.

11:09AM BL.00004 A new Ghost Fluid approach on non-graded adaptive cartesian grid for solving the Poisson equation with jump conditions enforced on an irregular interface , ASDIS HELGADOTTIR, FREDERIC GIBOU, UCSB — A poisson solver on a non-graded adaptive Cartesian grid, for the poisson equation with jump enforced at an irregular interface, is presented. A Ghost Fluid method, similar to that presented by Liu *et al.* (in JCP:160(2000), 151 - 178) is used, with two main differences: 1) the uniform grid is replaced with optimum quad tree (2D) and octree (3D) structures, which significantly saves computational time and memory usage, 2) the jump in the normal derivative is enforced analytically (or more so by a numerical integration of the analytical term) instead of being approximated with finite difference as done by Liu *et al.*. The method is simple and results in a positive definite matrix that can easily be solved with black box solvers.

11:22AM BL.00005 A new approach to computing open-boundary flows with SPH , S. MAJID HOSSEINI, JAMES FENG, Department of Chemical and Biological Engineering, University of British Columbia, Vancouver, BC V6T 1Z3, Canada — In Smoothed Particle Hydrodynamics (SPH) methods, incompressibility is typically imposed by a projection method. This entails an artificial Neumann boundary condition for the pressure Poisson equation, which is often inconsistent with physical conditions at inflow and outflow boundaries. For this reason open-boundary flows have rarely been computed using SPH. In this work, we demonstrate that the classical pressure boundary condition produces a numerical boundary layer that compromises the solution near the boundaries. We resolve this problem by utilizing a "rotational incremental pressure-correction scheme" with a consistent pressure boundary condition. We show that this scheme computes the pressure and velocity accurately near open boundaries, and extends the scope of SPH simulation beyond the usual closed and periodic boundary conditions.

11:35AM BL.00006 Exponential Time Integrator for Solving the Lattice Boltzmann Equations Based on a Spectral-Element Discontinuous Galerkin Approach , KALU CHIBUEZE UGA, CUNY - CCNY , MISUN MIN, ANL, TAEHUN LEE, CUNY - CCNY, PAUL FISCHER, ANL, CUNY - CCNY COLLABORATION, ANL COLLABORATION — I'll present a high-order time integration method for solving lattice Boltzmann equation (LBE). We use high-order spectral-element discretization in space based on discontinuous Galerkin approach and apply a Krylov subspace approximation for time-advancing. The semi-discrete form of the spectral-element discontinuous Galerkin (SEDG) method on the LBE brings us to an ODE of the form $\partial U/\partial t = -AU$ with initial condition$U(0) = U_0$ where $U$is a solution vector. A is a large sparse matrix based on a polynomial approximation order N. The solution of the equation is $U(t) = U_0 e^{-At}$. The explicit one-step method is based on the computation of matrix functions of the type $U(t+\delta) = U(t)e^{-A\delta}$. We project the matrix exponential and the solution vector onto a finite dimensional Krylov subspace K_m of order m. We use the Arnoldi algorithm to generate an orthogonal basis V_m and an Hessenberg matrixH_m for approximating $e^{-A\delta}U(t) \approx V_m e^{-H_m\delta}V_m^T U(t)$. We will study convergence of the exponential time integrator for possible use of larger time step with high-order m.We will demonstrate its efficiency and accuracy compared to the Runge-Kutta time-stepping methods.

11:48AM BL.00007 A Lattice-based Numerical Solution Of Collisional Boltzmann Equation , BOE GREEN, PRAKASH VEDULA, University of Oklahoma — Accurate prediction capabilities of nonequilibrium flow behavior are important for efficient design of a wide range of emerging technologies ranging from micro-scale flow devices and hypersonic re-entry vehicles. Challenges for nonequilibrium flow predictions are mainly due to the breakdown of the continuum flow field hypothesis and the corresponding classical continuum field equations (e.g. Navier-Stokes). To address these challenges we propose an efficient lattice-based computational method (called Collisional Lattice Boltzmann Method or cLBM) for description of non-equilibrium flows, based on fundamental Boltzmann kinetic theory and the underlying full collision operator, without the use of any equilibrium-based approximations. In this moment method, which is applicable for a wide range of Knudsen numbers, the contributions of the full collision operator to the evolution of moments are computed via multinomial expansions and analytical integrals using a discrete quadrature on the lattice while spatial transport is accounted for through a streaming process on the lattice. The underlying conservation laws and invariants of collisional relaxation are also preserved in our approach, along with rates of evolution of selected low order moments. Prediction capabilities of cLBM are demonstrated via good agreement between results obtained from cLBM and other approaches using Couette, Poiseulle and lid-driven cavity flows.

12:01PM BL.00008 Semi-implicit Unstructured Finite Element Lattice Boltzmann Equation Method for Incompressible Binary Fluids[1] , TAEHUN LEE, City College of New York — A semi-implicit finite element lattice Boltzmann equation method for incompressible binary fluids with large density and viscosity differences is proposed. The collision is treated implicitly and the intermolecular forcing terms are treated explicitly in order to achieve stability at high Reynolds number and avoid implicit treatment of the non-linear forcing terms. The characteristic Galerkin finite element approximation is adopted for the solution of the streaming, which provides geometric flexibility while retaining high-order accuracy. Unstructured body-fitted mesh enables mass conservation at the solid/liquid/gas triple contact line. The equilibrium contact angle is naturally imposed by the surface integral of the free energy. The proposed method is applied to several benchmark cases including drop sliding on patterned superhydrophobic surfaces.

[1]This work was supported by NSF DMS-0811046.

12:14PM BL.00009 Gas-kinetic BGK Schemes for 3D Viscous Flow[1] , JIN JIANG, YUEHONG QIAN, Shanghai University — Gas-kinetic BGK scheme developed as an Euler and Navier-Stokes solver is dated back to the early 1990s. There are now numerous literatures on the method. Here we focused on extending this approach to 3D viscous flow. Firstly, to validate the code, some test cases are carried out, including 1D Sod problem, interaction between shock and boundary layer. Then to improve its computational efficiency, two main convergence acceleration techniques, which are local time-stepping and implicit residual smoothing, have adopted and tested. The results indicate that the speed-up to convergence steady state is significant. The last is to incorporate turbulence model into current code with the increasing Reynolds number. As a proof of accuracy, the transonic flow over ONERA M6 wing and pressure distributions at various selected span-wise directions have been tested. The results are in good agreement with experimental data, which implies the extension to turbulent flow is very encouraging and of good help for further development.

[1]Supported by IRT0844, 10625210, 08ZZ43, 07PJ14041.

12:27PM BL.00010 A New Weighted-Integral Based High-Order Method , LI-JUN XUAN, JIE-ZHI WU[1], Peking University, Beijing China — A weighted-integral based scheme (WIBS) has been constructed where the integrals of the function weighted by test functions are recorded as degree of freedoms (DOFs). The time evolution of DOFs is by stable Runge-Kutta method from the weak form of the original equation. At the boundary of every two cells, the function values are interpolated from the DOFs of the neighboring cells to calculate flux and volumetric integral in the weak form. The basic idea is to increase the order of interpolation by increasing both interpolating cells and DOFs simultaneously. The interpolation on more cells permits the use of WENO to capture discontinuity, while more DOFs can shrink the size of the interpolating stencil. The compactness of the reconstruction can increase the accuracy (especially for short waves) and fully retain it at boundary. Many existing schemes (e.g., FV, FE, FD, DG, SV, SD, Hermiter, etc.) can be viewed as special subclasses of WIBS. For 1-D hyperbolic conservation law systems, a high stability is found and the order of accuracy is perfectly held. A WENO-WIBS scheme has also been constructed to capture the discontinuity successfully. The results of WIBS for various benchmark problems are compared with those of 5th-order WENO.

[1]Also at University of Tennessee Space Institute, USA

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BM Compressible Flows 200B

10:30AM BM.00001 Boundary Layer Entrainment and Combustion in a Transverse Jet in Supersonic Crossflow[1] , WILL HELTSLEY, MIRKO GAMBA, GODFREY MUNGAL, RONALD HANSON, Stanford University — Recent experiments have reconstructed the 3D structure of the combustion regions around burning jets-in-supersonic-crossflow using OH-PLIF in multiple orthogonal planes. A sonic hydrogen jet with a momentum-flux ratio of $J = 4$ was injected normally into high temperature air crossflows at two freestream conditions: (a) T= 1250 K, M= 2.7, P= 25 kPa, and (b) T= 1500 K, M= 2.4, P= 25 kPa. These freestream conditions produce stable combustion in the shear layer of the jet at J = 4. The T = 1500 K case also produces an intermittently reacting recirculation region in the shock foot region at the front of the jet, and a highly reactive boundary layer. The boundary layer ignites in the recirculation region, wraps around the jet at the base of the jet's bow shock cell and develops downstream where it is entrained into the jet wake. This previously unreported fuel entrainment and ignition in the near field boundary layer may be a significant mechanism for ignition and flame stabilization in the far field of such transverse jets.

[1]Sponsored by AFOSR/MURI, J Tishkoff, Technical Monitor.

10:43AM BM.00002 Simulation of Scramjet Fuel Injection Using a Hybrid RANS/LES Approach , DAVID PETERSON, GRAHAM CANDLER, University of Minnesota — A methodology is presented for the simulation of realistic scramjet combustor sections at actual operating conditions. A flow solver with an unstructured grid framework is used such that grids can be constructed for complex geometries. Numerics are chosen for robustness and reliability, while not being overly dissipative as to overwhelm the actual flow physics. The flow around a scramjet fuel injector is highly unsteady and dominated by coherent, large-scale structures. The goal is to be able to capture the key physics of the flow field, without resolving all of the detail of the flow. In regions of the flow where unsteady phenomenon are important, such as where mixing occurs or in massively separated regions, LES is used. RANS is used in regions of attached flow, such as boundary layers, where RANS performs well. In addition, RANS is also used as a wall model for the LES regions of the flow. This allows for realistic Reynolds number to be simulated at reasonable cost. Comparisons are made between simulations and available experimental data.

10:56AM BM.00003 A Computational Study of Hypervelocity Boundary Layer Stability by Means of Carbon Dioxide Injection , ROSS WAGNILD, GRAHAM CANDLER, University of Minnesota — An experimental method for introducing carbon dioxide as a means of stabilizing a hypervelocity boundary layer over a slender bodied vehicle is investigated through the use of numerical simulations. The motivation for this concept is derived from the results of previous experiments and calculations which demonstrated the ability of fluid flows in chemical and thermal non-equilibrium to absorb energy from acoustic disturbances, particularly flows involving carbon dioxide. In the current study, the computational model is a five-degree half-angle cone with a single row of injector ports. The effect of the injection and the transition location are determined by solving the parabolized stability equations and using the semi-empirical eN method. The preliminary results show a delay in the transition location as compared to a cone without injection under the same flight conditions.

11:09AM BM.00004 Boundary Layer Transition on Elliptic Cones in Hypersonic Flow[1] , MATTHEW BARTKOWICZ, PRAMOD SUBBAREDDY, GRAHAM CANDLER, University of Minnesota — We are studying transition to turbulence at hypersonic conditions using a combination of parabolized stability equations and direct numerical simulations. In this work we perform DNS of acoustic disturbances interacting with the Mach 8 flow over a 4:1 elliptic cross-section cone to model the experiments of Huntley and Smits (2000). A fourth-order accurate low-dissipation numerical method is used, with large-scale body-fitted grids to obtain accurate results. A spectrum of planar random phase acoustic disturbances are introduced in the free-stream with amplitudes consistent with experiment. We observe instability growth and non-linear behavior similar to that observed in the filter Rayleigh scattering flow visualization taken at Princeton.

[1]This work was supported by NASA.

11:22AM BM.00005 Unsteadiness of low-Reynolds-number shock / boundary layer interactions , VENKATESWARAN NARAYANASWAMY, NOEL CLEMENS, The University of Texas at Austin — Low Re shock wave / boundary layer interactions (SWBLI) generated by compression ramps in a Mach 3 flow are studied experimentally. Ramp angles ranging from 16° to 24° are used to produce separated flows of varying strength (or length-scale). The upstream boundary layers are turbulent with $Re_\theta \approx 3000$ - 5500. This study aims to understand the dominant mechanisms that drive the low-frequency oscillations of the separation bubble as a function of separated flow scale. The organization of the separation bubble is studied using simultaneous wall pressure measurements underneath the separation bubble. Coherence and magnitude of cross correlation between pressure fluctuations in the intermittent region and inside separation bubble is found to be between those of low Re impinging SWBLI and high Re SWBLI. Simultaneous PIV in the upstream boundary layer and wall pressure measurements are used to characterize the influence of the incoming boundary layer fluctuations on the separation bubble dynamics. Furthermore, correlations between the velocity within the separated flow and wall pressure fluctuations are used to study instabilities intrinsic to the separated flow.

11:35AM BM.00006 Helium injection in a turbulent boundary layer at hypersonic speeds , PARTHAV DESAI, DIPANKAR SAHOO, ALEXANDER SMITS, Department of Mechanical and Aerospace Engineering, Princeton University — Low level of helium injection into a transitional or turbulent boundary layer can have substantial effect on the boundary layer structure, as shown by Auvity, Etz, Smits (Physics of Fluids, Vol 13 no 10, 2001) for the flow over a flat plate at Mach 8. Here, we report the results of additional experiments to study the effects of helium injection on the turbulence statistics and the wall heat flux. We use PIV to obtain the velocity fields, and infrared thermography to derive the wall heat transfer distribution. The helium is injected through a slot just downstream of the trip wire, and the ratio of the momentum flux of the helium to that of the freestream flow is varied from 0 to about 0.16, as in the case of Auvity et al. Supported under NASA Grant NNX08AB46A, Program Manager Catherine McGinley.

11:48AM BM.00007 Measurement of Fluctuating Wall Pressures Beneath a Supersonic Turbulent Boundary Layer , STEVEN BERESH, JOHN HENFLING, RUSSELL SPILLERS, BRIAN PRUETT, Sandia National Laboratories — Accurate measurement of fluctuating wall pressure spectra beneath a supersonic turbulent boundary layer has proven elusive, such that a compilation of past efforts exhibits an alarming degree of scatter and hinders the development of engineering models. Recent experiments conducted in Sandia's Trisonic Wind Tunnel up to Mach 3 have provided wall pressure data to frequencies exceeding 100 kHz to help reconcile conflicts in the historical data. Data were acquired using piezoresistive silicon pressure transducers effective at low- and mid-range frequencies, then supplemented by piezoelectric quartz sensors capable of detecting very high frequency events. The two sensor types were dynamically calibrated against a condenser microphone reference standard, then combined into a single curve describing the wall pressure spectra. Such spectra show that an increase in Mach number produces a reduction in the normalized magnitude, though the shape of the spectra remain similar; Reynolds number effects were detectable but considerably smaller for the range of test conditions. Results are compared with historical data and consequences of the measurement limitations are discussed.

12:01PM BM.00008 LES and DNS of Shock-Boundary Layer Interactions. , AVINASH JAMMALAMADAKA, ZHAORUI LI, FARHAD JABERI — Large-eddy simulations (LES) of an incident oblique shock wave interacting with a flat-plate supersonic turbulent boundary layer at various flow/shock conditions are performed and the results are compared with the direct numerical simulation (DNS) data. The objectives are to evaluate the performance of compressible subgrid-scale (SGS) models in shock-turbulence flow regions and to study the effects of shock angle, Mach number and other parameters on the shock-boundary layer interactions. The filtered compressible Navier-Stokes equations are solved with a seventh-order Monotonicity-Preserving scheme for the Euler fluxes and a sixth-order compact scheme for the viscous terms. Comparison of DNS and LES results reveal the significance of the SGS model in supersonic boundary layer flow, particularly in the shock-turbulence regions. Due to its excessive dissipative nature, the standard Smagorinsky and gradient type models are found to predict a significantly larger mean separation bubble size when compared to the DNS data. In contrast, the predicted results by the scale-similarity, mixed or dynamic Smagorinsky models are found to be in reasonably good agreement with the DNS. Similar trends are observed for all the major flow variables. In general, the dynamic models, though computationally expensive, are found to generate better results when compared to other models.

12:14PM BM.00009 Upstream Boundary Condition Sensitivity of the Shock-Boundary Layer Interaction , DAVID HELMER, TONKID CHANTRASMI, CHRIS ELKINS, GIANLUCA IACCARINO, JOHN EATON, Stanford University — A low aspect ratio Mach 2.1 wind tunnel with a 20 $^\circ$ compression wedge is being used to validate uncertainty quantification techniques for CFD. The tunnel is operated continuously, with a mass flow rate of $\sim$0.7kg/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 fluctuation applied in the CFD. The compression wedge causes an oblique shock to form, resulting in flow separation at the base of the wedge. Pressure data are measured using a closely-spaced array of taps near the base of the wedge to map the 2D footprint of the shock. These data show that the flow is only weakly three dimensional. PIV measurements are taken throughout the field, with a focus on the shock-boundary layer interaction at the base of the compression wedge. The field of measurement also includes the location where the shock impinges on the opposite wall, where another separation occurs. Measurements are compared to various CFD simulations to see how different modeling assumptions affect the result and to evaluate the validity of CFD uncertainty quantification techniques.

12:27PM BM.00010 MHD Flow Control of Oblique Shock Induced Separation , JOHN EKATERINARIS, FORTH/IACM and University of Patras — The effect of magnetic fields on the flow of conducting fluids is well known. There is current interest to exploit the body forces induced by they magnetic field for the control of separation, transition, and turbulence. The wave structure of ionized gas high speed flows under the influence of magnetic fields, which contains slow, Alfvιn, and fast waves, is more involved than the waves of ordinary gas dynamics. High resolution is required to accurately compute interactions of complex wave discontinuities and smooth flow features. Numerical solutions of the ideal and viscous magnetohydrodynamic (MHD) equations are obtained with a high order accurate shock capturing scheme. The numerical method was validated to ensure that it provides crisp resolution of discontinuities, it maintains high-order accuracy for the smooth parts of the flow, it preserves numerical stability, and eliminates nonphysical features that result from the violation of the divergence-free condition for the magnetic field. The numerical method was then applied to simulate separation control at the interaction region of an oblique shock with a laminar boundary layer under the influence of magnetic fields. It was found that magnetic fields can significantly reduce shock induced separation that appears upstream of the interaction region and can cause rapid transition to turbulence.

Sunday, November 22, 2009 10:30AM - 12:27PM –

Session BN Experimental Techniques II: Near Wall Flows 200C

10:30AM BN.00001 Wall shear measurements of turbulent flow over backward facing step , NGUYEN THIEN DUY, WELLS JOHN CRAIG, Ritsumeikan University, Japan, NGUYEN CHUONG VINH, Monash University — This paper is aim to generalize our recent developments of an extension of particle image velocimetry (PIV) measurement technique, named as Interfacial PIV (noted as IPIV). It enables us to deal with near-wall flows over curved walls by means of conformal transformation. In addition, if the displacement of tracer in wall-normal direction is less than its diameter, IPIV could instantaneously and precisely measure the wall shear gradient as well as draw out the tangential velocity profile. Our integration of measured velocity gradients upward from wall could provide a continuous profile with single pixel resolution. In this communication, we extend our wall shear measurement technique to stereo-PIV setup. Validation of IPIV wall shear measurement to 2-dimensional and 3-dimensional synthetic images of turbulent flows over a wavy bed is performed. In practice, experiment with backward-facing step (BFS) at a low Reynolds number of 2800 is examined. A system of two two-component PIV (2CPIV) coupled with a stereo-PIV (SPIV) is conducted to inquire the flow structure. Applications of IPIV to our 2CPIV and SPIV experimental images are also described.

10:43AM BN.00002 Measurements of mechanical torque acting on a model wind turbine[1] , HYUNG-SUK KANG, Johns Hopkins University, RAUL B. CAL, Portland State University, JOSE LEBRON-BOSQUES, LUCIANO CASTILLO, Rensselaer Polytechnic Institute, CHARLES MENEVEAU, Mechanical Engineering and Center for Environmental and Applied Fluid Mechanics, Johns Hopkins University — A high precision torque converter is developed and calibrated to measure torques on model wind-turbine generators (small DC motor). By multiplying the measured torque and rotor angular velocity, a direct measurement of the extracted power from the wind turbine can be obtained. This direct method is more advantageous compared to the electrical power measured from the model generator, since it avoids contamination by internal friction, and electric and magnetic losses. The torque converter is mounted on a model wind turbine in a 3 by 3 array (3 rows by 3 columns) of wind turbines in the Corrsin wind tunnel. The power coefficients are obtained as a function of the tip speed ratio. Significant difference between the electrical and mechanical powers is observed, which emphasizes on the importance in using the direct mechanical power measurement. Also, the extracted power by the model turbine is found to be comparable with the power estimated from the kinetic energy flux measured using SPIV.

[1]Supported by the NSF (CBET-0730922).

10:56AM BN.00003 Measurement of the Wall Shear Stress Field Created by a Supersonic Impinging Microjet , BRIDGET SCHABRON, JONATHAN NAUGHTON, University of Wyoming, FARRUKH ALVI, Florida State University — The study of wall shear stress beneath a supersonic impinging microjet was carried out using oil film interferometry. The measurement of wall shear stress in a microjet is an interesting flow both for its practical applications as well as the challenges the surface flow poses. The wall shear stress measurement challenges for this flow included the radially symmetric wall shear stress field, the very high gradients of wall shear stress, optical access issues, and the significant temperature variations on the surface that affect the oil's viscosity. Measurements were made for various jet height above the plate h to jet diameter D ratios and for various pressure ratios. Analysis of the resulting interferograms and the challenges posed by this particular geometry are discussed. The results demonstrate the capability of oil film interferometry, particularly its dynamic range, for measuring wall shear stress in complex flow.

11:09AM BN.00004 The diagnostic plot - a new way to appraise turbulent boundary layer data , P.H. ALFREDSSON, R. OERLUE, Linne FLOW Centre, KTH Mechanics — Most turbulent boundary layer data are obtained with hot-wire anemometry which gives access both to the mean (U) and turbulence intensity (u') distributions of the streamwise velocity. Comparisons between different measurements strongly depend on the accuracy of the determination of the friction velocity (u_τ) and for comparisons in the near wall region the determination of the wall position is crucial. If u' is plotted as function of U, where both quantities are normalized by the free stream velocity (U_∞) (hereafter called the *diagnostic plot*), any uncertainties in the wall position and u_τ are avoided when comparing different cases. For a given Re all such distributions should fall on top of each other if u' and U are accurately measured and the measurements are made in standard zero pressure gradient turbulent boundary layers. Close to the wall ($y^+ < 10$) the distribution is nearly self similar and independent of Re. Moreover the distribution should be linear with a constant slope (~ 0.40) up to, at least, $y^+ = 3$ (corresponding to $U/U_\infty > 0.1$ for typical laboratory experiments) and thereafter the slope should decrease. The diagnostic plot clearly indicates at what position the measured values show a wall interference effect. Also in the outer region the distributions at different Reynolds numbers overlap, and the diagnostic plot has the interesting property that both the inner ($y^+ < 10$) and outer regions can be made to collapse in the same plot.

11:22AM BN.00005 Simultaneous drag and flow measurements of Olympic skeleton athletes , YAE EUN MOON, DAVID DIGIULIO, RPI, STEVE PETERS, US Bobsled & Skeleton Federation, TIMOTHY WEI, RPI — The Olympic sport of skeleton involves an athlete riding a small sled face first down a bobsled track at speeds up to 130 km/hr. In these races, the difference between gold and missing the medal stand altogether can be hundredths of a second per run. As such, reducing aerodynamic drag through proper body positioning is of first order importance. To better study the flow behavior and to improve the performance of the athletes, we constructed a static force balance system on a mock section of a bobsled track. Athlete and the sled are placed on the force balance system which is positioned at the exit of an open loop wind tunnel. Simultaneous drag force and DPIV velocity field measurements were made along with video recordings of body position to aid the athletes in determining their optimal aerodynamic body position.

11:35AM BN.00006 Near Wall measurement in Turbulent Flow over Rough Wall using microscopic HPIV[1] , SIDDHARTH TALAPATRA, Johns Hopkins University, JIARONG HONG, Johns Hopkins Unversity, JOSEPH KATZ, Johns Hopkins University — Using holographic PIV, 3D velocity measurements are being performed in a turbulent rough wall channel flow. Our objective is to examine the contribution of coherent structures to the flow dynamics, momentum and energy fluxes in the roughness sublayer. The 0.45mm high, pyramid-shaped roughness is uniformly distributed on the top and bottom surfaces of a 5X20cm rectangular channel flow, where the Re_τ is 3400. To facilitate recording of holograms through a rough plate, the working fluid is a concentrated solution of NaI in water, whose optical refractive index is matched with that of the acrylic rough plates. The test section is illuminated by a collimated laser beam from the top, and the sample volume extends from the bottom wall up to 7 roughness heights. After passing through the sample volume, the in-line hologram is magnified and recorded on a 4864X3248 pixels camera at a resolution of 0.74μm/pixel. The flow is locally seeded with 2μm particles. Reconstruction, spatial filtering and particle tracking provide the 3D velocity field. This approach has been successfully implemented recently, as preliminary data demonstrate.

[1]Sponsored by ONR

11:48AM BN.00007 Performance of a Photonic Wall Shear Stress Sensor[1] , ULAS AYAZ[2], TINDARO IOPPOLO[3], VOLKAN OTUGEN[4], Southern Methodist University — The performance of a photonic wall shear stress sensor prototype based on the so-called whispering gallery modes (WGM) of polymeric microspheres is investigated in steady and unsteady flows. In this sensor, the shear force due to the fluid flow is transmitted to a Polydimethylsyloxane sphere of several hundred microns in diameter which serves as the sensor. The corresponding optical resonance (WGM) shifts are monitored to determine the wall shear stress. Sensor performance for dynamic range, resolution and bandwidth are studied analytically, and validated experimentally. The validation experiments for the prototype sensor with measurement area of 1mm^2 are made in a two-dimensional channel flow and in an acoustic plane wave tube. These measurements indicate a shear stress resolution of $\sim 10^{-3}$Pa and a dynamic range of $\sim$100dB for the prototype. The PDMS sphere used in the prototype has a base-curing-agent ratio of 40:1. Different sensitivities and measurement ranges can be obtained using different PDMS mixing ratios.

[1]Research supported by NSF
[2]Mechanical Engineering Dept., Doctoral student
[3]Mechanical Engineering Dept., post-doctoral fellow
[4]Mechanical Engineering Dept., Professor

12:01PM BN.00008 Simultaneous Reference- and Signal-Imaging for Cancellation of Unsteady Motion in Pressure-Sensitive Paint Measurement , KENSUKE MIYAMOTO, TAKESHI MIYAZAKI, The University of Electro-Communications, HIROTAKA SAKAUE, JAXA — Simultaneous reference- and signal-image acquisition for pressure-sensitive paint (PSP) measurement is presented in this paper. The system to acquire these images consists of a two-color PSP and a high-speed CCD camera. The two-color PSP provides a reference and a signal luminescence separated by their luminescent peak wavelengths of 520 nm and 620 nm, respectively. The reference luminescent image is acquired through a green filtered CCD, while the signal image through a red filtered CCD. This system can cancel a non-uniform illumination for exciting two-color PSP as well as a non-uniform image acquisition of a CCD camera due to the location between the camera and the testing object. This system can be applied to a PSP measurement, which includes translation, vibration, and deformation of the testing object. Our measurement system is thus advantage from a conventional PSP measurement, which requires stationary location among the testing object, camera, and illumination source. Unsteady motion of a plate coated with our two-color PSP is included as a demonstration. The developed system captured the unsteady motion as well as gas impingement on the coated plate with camera frame rate of 300 Hz.

12:14PM BN.00009 Global Visualization in Water using AnodizedAluminum PressureSensitive Paint and Dissolved Oxygen as Tracer , TATSUYA OZAKI, HITOSHI ISHIKAWA, Tokyo University of Science, HIROTAKA SAKAUE, JAXA — We have developed anodized-aluminum pressuresensitive paint (AA-PSP) for flow visualization in water using dissolved oxygen as a tracer. Developed AA-PSP is characterized using water calibration setup by controlling a dissolved oxygen concentration. It is shown that the developed AA-PSP gives 4.0 percent change in luminescence per 1 mg/l of oxygen concentration. This AA-PSP is applied to visualize flows in a water tunnel. Oxygen concentrations of the water tunnel and the dissolved oxygen are 9.5 mg/l and 20 mg/l, respectively. We can capture horseshoe vortices over the base of 10 mm cylinder by using this technique at Reynolds number of 1000 and a water speed of 100 mm/s, respectively. Unlike conventional tracers such as ink, milk, and fluorescent dyes, this visualization technique gives flow information on the AA-PSP coated surface without integrating flows between the AA-PSP and an optical detector. Because of using dissolved oxygen as a tracer, it holds the material properties of testing water except for the amount of oxygen. The tracer does not interfere with optical measurements and it does not contaminate the testing water. A conventional visualization technique using milk as a tracer is also employed for comparison.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BP Instability: Boundary Layers II 200D

10:30AM BP.00001 Modal description of optimal internal streaks in a Falkner-Skan Boundary Layer , JOSE SANCHEZ-ALVAREZ, MARIA HIGUERA, JOSE MANUEL VEGA, ETSI Aeronauticos. UPM — Understanding the growing disturbances in boundary layers is of crucial interest for numerous engineering applications. Here we examine the optimal streaky perturbations (which maximize energy growth) in a wedge flow boundary layer. These 3D perturbations are governed by a system of equations obtained by linearizing the 3D Navier-Stokes equations around the base flow given by the Falkner-Skan similarity solution. Based on an asymptotic analysis of this system near the free stream and the leading edge singularity, we show that optimal streaks can be described in terms of a single streamwise-growing solution of the linearized equations, which is associated with an eigenvalue problem first formulated in this context by Tumin (Phys. Fluids 13, 5, (2001)). Such a solution may be regarded as an internal spatially-unstable mode, in analogy with the usual eigenmodes of standard linear stability theory. An important consequence of this result is that the optimization procedure heretofore used to define optimal streaks is not necessary. Comparison with previous results in A.Tumin and D.E. Ashpis AIAA (2003) show excellent agreement.

10:43AM BP.00002 RNS streak description , CARLOS MARTEL, JUAN ANGEL MARTIN, ETSI Aeronauticos, Universidad Politecnica de Madrid, SPAIN — We use the Reduced Navier-Stokes (RNS) equations for the simulation of the nonlinear evolution of streaks in a flat plate boundary layer. The RNS are asymptotically derived from the Navier Stokes equations for $Re \gg 1$, and they are appropriated for flow configurations with one slow scale and two short scales. We derive the RNS with the appropriate boundary conditions for the simulation of the spatially growing streaks, comment the details of the numerical method used, and compare our 3D streak simulations with the results present in the literature. The presented RNS scheme for computing nonlinear streaks is much faster than full 3D DNS computations, and does not exhibit the numerical instabilities present in previous nonlinear PSE calculations.

10:56AM BP.00003 Optimal Disturbances and Receptivity of 3D Boundary Layers[1] , DAVID TEMPELMANN, KTH Mechanics, ARDESHIR HANIFI, FOI Sweden & KTH Mechanics, DAN HENNINGSON, KTH Mechanics — We will present spatial optimal disturbances in a Falkner-Skan-Cooke boundary layer and illuminate how these can be used to determine the receptivity of crossflow vortices to freestream disturbances. Optimal disturbances, which are obtained by solving a parabolized set of equations, initially take the form of vortices tilted against the direction of the mean crossflow shear. Further downstream they evolve into bended streaks and finally into crossflow disturbances. A large potential for initial non-modal growth becomes apparent where both the lift-up effect and the Orr-mechanism are identified as responsible physical mechanisms. We inquire if non-modal growth is related to a receptivity mechanism for modal instabilities in 3D boundary layers. We therefore use continuous modes from the Orr-Sommerfeld/Squire spectrum as a model for freestream turbulence and project them onto initial optimal disturbances in order to obtain receptivity coefficients. A parametric study concerning optimal growth and receptivity will be presented as well as a comparison to existing DNS and experimental data.

[1]This work is supported by the European Commission through the FP6 project TELFONA (Contract No AST4-CT-2005-516109).

11:09AM BP.00004 Scaling of Transient Growth of Instability Induced by a Periodic Array of Roughness Elements in a Blasius Boundary Layer , PHILIPPE LAVOIE, University of Toronto, AHMED NAGUIB, Michigan State University, JONATHAN MORRISON, Imperial College — The receptivity of laminar boundary layers to three-dimensional perturbations and the ensuing evolution of the transient growth of disturbances have attracted much attention in recent years. The motivation for the present study is related to the development of a reduced-order model and estimator for the closed-loop control of transient growth instabilities in a laminar boundary layer. We focus here on the scaling characteristics of the streamwise component of the disturbance energy. Starting from the linearized boundary layer equations and given assumptions with respect to the boundary layer receptivity to perturbations, we derive scaling arguments for the evolution of the disturbance energy. These are examined against experimental data, which were obtained by inducing transient growth in a Blasius boundary layer in the wind tunnel using spanwise-periodic arrays of cylindrical roughness elements with different geometrical parameters. It is found that the growth and decay region of the energy evolution scale differently. The dynamical implications of the scaling presented here are discussed from the point of view of model reduction for flow control.

11:22AM BP.00005 Toward the foundation of a global modes concept , ANATOLI TUMIN, The University of Arizona — Recent progress in using global modes for the analysis of a variety of complex (and "simple") flows led to their applications in flow control. The progress in computational capabilities brings this advanced technique to common practice in studies of flow perturbations. However, the formulation of global eigenvalue problems is accompanied by uncertainties in the choice of boundary conditions. Today, the choice of boundary conditions has a heuristic nature, and this provokes questions regarding the suitable formulation of the eigenvalue problems. A simple model can help us to understand the effect of the upstream and downstream boundary conditions on the eigenvalues and eigenfunctions. I consider the "box formulation" in a uniform flow. In the limit of an infinite domain in the y-direction, the problem is reduced to a system of ODEs with constant coefficients using the Fourier transform in y. There are boundary layers in the vicinity of the upstream and downstream boundaries. The pressure wave penetrates upstream at a distance about the characteristic scale of the perturbation in the y direction. Discussion of Dirichlet and Neumann boundary conditions for the model problem is accompanied by comparisons with available publications for boundary layer flows. The model helps to understand the spectrum features in published results.

11:35AM BP.00006 How many unstable modes are in high-speed boundary layers?[1] , ALEXANDER FEDOROV, Moscow Institute of Physics and Technology, ANATOLI TUMIN, The University of Arizona — L. Mack (1969) carried out the inviscid stability analysis of high-speed boundary layers, and he found that in addition to the unstable mode having a viscous nature, there are other unstable discrete modes associated with acoustic perturbations "trapped" inside the boundary layer. In contemporary stability analysis of high-speed boundary layers at finite Reynolds numbers, two discrete modes are identified: the 1st and 2nd modes. However, at high Mach numbers and finite Reynolds numbers, the discrete spectrum of normal modes has only one eigenvalue that is meandering in the complex plane (the wave number or the frequency, depending on the stability framework) that corresponds to an unstable perturbation. In the present work, we illustrate how the synchronism and branching of discrete modes can lead to a spectrum with one or two unstable discrete normal modes. One has to be aware of this phenomenon and to keep in mind the ambiguity associated with the terminology of the "1st mode" and "2nd mode."

[1]This work was supported by the AFOSR/NASA National Center for Hypersonic Research in Laminar-Turbulent Transition and by AFOSR grant No. FA95550-08-1-0322.

11:48AM BP.00007 Large Eddy Simulation of Transitional Boundary Layer[1] , TARANEH SAYADI, PARVIZ MOIN, CTR, Stanford University — A sixth order compact finite difference code is employed to investigate compressible Large Eddy Simulation (LES) of subharmonic transition of a spatially developing zero pressure gradient boundary layer, at $Ma = 0.2$. The computational domain extends from $Re_x = 10^5$, where laminar blowing and suction excites the most unstable fundamental and sub-harmonic modes, to fully turbulent stage at $Re_x = 10.1 \times 10^5$. Numerical sponges are used in the neighborhood of external boundaries to provide non-reflective conditions. Our interest lies in the performance of the dynamic subgrid scale (SGS) model [1] in the transition process. It is observed that in early stages of transition the eddy viscosity is much smaller than the physical viscosity. As a result the amplitudes of selected harmonics are in very good agreement with the experimental data [2]. The model's contribution gradually increases during the last stages of transition process and the dynamic eddy viscosity becomes fully active and dominant in the turbulent region. Consistent with this trend the skin friction coefficient versus Re_x diverges from its laminar profile and converges to the turbulent profile after an overshoot. 1. Moin P. *et. al.* Phys Fluids A, **3**(11), 2746-2757, 1991. 2. Kachanov Yu. S. *et. al.* JFM, **138**, 209-247, 1983.

[1]Supported by the PSAAP program of DOE

12:01PM BP.00008 Effects of passive porous walls on hypersonic boundary layers[1] , SHARON STEPHEN, VIPIN MICHAEL, University of Birmingham — We consider the effect of a passive porous wall on the first mode of a hypersonic boundary layer on a sharp slender cone. A theoretical stability analysis is used for large Mach number and large Reynolds number which includes the effect of curvature and of the attached shock. The formulation considers the scales appropriate to the first mode which is associated with Tollmien- Schlichting waves and this results in a triple-deck structure. The flow in the hypersonic boundary layer is coupled to the flow in the porous layer. We consider the porous layer on the cone surface to be a sheet perforated with cylindrical blind holes of equal spacing[2]. The linear stability analysis results in an eigenrelation, relating the streamwise wavenumber and the frequency of the disturbances. Neutral solutions will be presented, indicating a destabilizing efect of the porous wall. Spatial growth rates obtained will demonstrate that the porous wall leads to a significant increase in disturbance growth rates. In addition, the effect of nonlinearity is considered.

[1]This effort is partially sponsored by the Air Force Office of Scientific Research, Air Force Material Command, USAF, under grant number FA8655-08-1-3044.

[2]Fedorov, A. V., Malmuth, N. D., Rasheed, A. and Hornung, H. G. *AIAA J.* **39**, 605 (2001).

12:14PM BP.00009 Investigation of Transition Initiated by a Wave Packet in a Hypersonic Cone Boundary Layer[1] , JAYAHAR SIVASUBRAMANIAN, HERMANN FASEL, University of Arizona — Direct Numerical Simulations (DNS) are performed to investigate the linear and nonlinear transition regime of a hypersonic boundary layer on a sharp circular cone at zero angle of attack. In a natural transition scenario a broad disturbance spectrum is excited by freestream disturbances leading to complex wave interactions. Therefore, in order to understand the natural transition process in hypersonic cone boundary layers, the flow was pulsed through a hole on the cone surface to generate a wave packet with a wide range of disturbance waves. First, DNS of a linear three-dimensional wave packet was performed and results are compared to linear stability theory (LST). A good agreement was found to exist between DNS and LST results. Then to study the nonlinear regime, DNS of a nonlinear wave packet was performed. The wall pressure disturbance spectrum of the nonlinear wave packet indicated the presence of fundamental and subharmonic resonance mechanisms.

[1]Funded by the Air Force Office for Scientific Research under grant FA9550-08-1-0211.

12:27PM BP.00010 Hypersonic Boundary Layer Stabilization Through Chemical Energy Absorption , HEATH JOHNSON, GRAHAM CANDLER, University of Minnesota — A number of studies have been performed looking at the effects of surface blowing or suction on boundary layer stability. This situation can arise in the case of forced mass flow through a porous surface or in the case of an ablative surface material which absorbs energy and releases gas into the boundary layer. Here we investigate the effects of blowing or suction in high-enthalpy, hypersonic flows where the injected gas will not only mix with the free-steam gas, but may also become vibrationally excited and chemically react both in the steady mean flow and in the unsteady disturbances. The effect of chemical and vibrational energy exchange on boundary layer stability is investigated in numerical simulations through application of the parabolized stability equations.

Sunday, November 22, 2009 10:30AM - 12:40PM –

Session BQ Instability: Interfacial and Thin-Film II 200E

10:30AM BQ.00001 Suppressing van der Waals rupture of thin films by imposed shear flow[1] , MICHAEL DAVIS, STEPHEN DAVIS, Department of Engineering Sciences and Applied Mathematics, Northwestern University — It is known that thin viscous films subject to attractive van der Waals forces will rupture in finite time due to a long-wave instability. We have studied the effects of applying a shear stress to the free surface of a film on a substrate, and found that it stabilizes perturbations in the direction of shear flow, thereby retarding rupture, or even suppressing rupture entirely for shear stress above a critical value. Perturbations orthogonal to the shear flow are not stabilized, and therefore a unidirectional shear will not prevent rupture in a three- dimensional viscous film. However, it may be possible to stabilize the film in all directions through the application of a rotating shear stress.

[1]M. Davis is grateful for the support of the NSF through the RTG program. M. Davis and S. H. Davis were partially supported by a grant from NSF/CMMI.

10:43AM BQ.00002 ABSTRACT WITHDRAWN –

10:56AM BQ.00003 Shear-induced suppression of rupture in two-layer thin liquid films , SREERAM K. KALPATHY, LORRAINE F. FRANCIS, SATISH KUMAR, University of Minnesota — The effect of shear on the rupture of two stratified thin liquid films confined between parallel plates and subject to van der Waals forces is examined in this work. Lubrication theory is applied to derive a one-dimensional nonlinear evolution equation for the height of the liquid-liquid interface. Linear stability analysis reveals that the real part of the growth rate and the wavelength of the fastest growing interfacial disturbance are unaffected by shear. However, the growth rate has an imaginary part which is non-zero in the presence of shear, indicating the existence of traveling waves. Nonlinear simulations of the interface behavior on homogeneous surfaces show that shear delays interfacial rupture, and suppression of rupture occurs beyond a critical shear rate. Propagation of traveling waves along the interface, and subsequent weakening of van-der-Waals-driven dewetting, is found to be the cause of the rupture delay. Analysis of flow on chemically heterogeneous surfaces also suggests a delay in interfacial rupture in the presence of shear. The problem studied here can serve as an idealized model for the lithographic printing process, and the results suggest that in the regime of shear rates relevant to printing, mechanisms of emulsification of one liquid into the other can be understood without incorporating shear. However, shear could be relevant in other physical systems such as microfluidic flows.

11:09AM BQ.00004 Spontaneous rupture of thinning liquid films with Plateau borders[1] , ANTHONY ANDERSON, Northwestern University, LUCIEN BRUSH, University of Washington, STEPHEN DAVIS, Northwestern University — Spontaneous film rupture from van der Waals instability is investigated in 2D. A thin liquid film between adjacent bubbles in a foam has finite length, curved boundaries (Plateau borders), and a drainage flow from capillary suction that causes thinning. A full linear stability analysis of this thinning film shows that rupture occurs once the film has thinned to *tens* of nanometers. Whereas, in an unbounded, quiescent, flat free film, rupture occurs when the thickness is *hundreds* of nanometers. Finite length, Plateau borders and flow are all found to contribute to the stabilization. The drainage flow leads to several distinct qualitative features as well. In particular, unstable disturbances are advected by the flow to the edges of the thin film. As a result, the edges of the film close to the Plateau borders are more susceptible to rupture that the center of the film.

[1]National Science Foundation, Grant Nos. CMMI-0827101 (LNB) and DMS-0636574 (AMA).

11:22AM BQ.00005 Plateau border adjustment in non-equilibrium foams[1] , MICHAEL GRATTON, STEPHEN H. DAVIS, Northwestern University — For foams without surfactants, changes occur in the Plateau border regions at the corner of nearly-polygonal bubbles three orders of magnitude faster than the thinning of lamellas. We describe the relaxation of an asymmetric Plateau border to symmetry in a two dimensional foam and compare the results to the Stoker-Hosoi hyperbolic coordinate theory for arid foams. These results are used to write a lumped-element model to describe the moderate timescale evolution of a foam, away from the time of lamella rupture, but slower than the timescale of local Plateau border adjustment.

[1]Supported by NSF/RTG and NSF/CMMI

11:35AM BQ.00006 Dynamics of thin annular films with electrokinetic effects[1] , DEVIN CONROY, Imperial College London, RICHARD CRASTER, University of Alberta, DEMETRIOS PAPAGEORGIOU, OMAR MATAR, Imperial College London — The evolution of an electrolyte in a uniform cylindrical tube in the presence of an applied electric field is investigated. A thread of a perfectly conducting fluid occupies the core of the tube. We derive an evolution equation for the interfacial position in the limit where the annular film is thin. This equation accounts for electrostatic and electrokinetic effects, and is characterised by an electric capillary number, a dimensionless Debye length and a ratio of interface to wall electrostatic potentials. We explore the effect of electrokinetics on the interfacial dynamics using a linear stability analysis and transient numerical simulations. The electrokinetics are shown to either stabilise or destabilise the film and, in the former case, causes the film to rupture in finite time. In this case, the time to touch down scales as time to the one-third and the final film shape undergoes either a ring or line-like rupture.

[1]EPSRC Grant number EP/E027393

11:48AM BQ.00007 Deformation of fluid free surfaces driven by high frequency vibration , MING TAN, JAMES FRIEND, Monash University, OMAR MATAR, Imperial College London, LESLIE YEO, Monash University — Formation of surface waves on a free surface of a thin fluid layer driven by high frequency (f $\approx$ 20 MHz) surface acoustic waves (SAWs) is investigated, both numerically and experimentally. The SAWs are transmitted along a surface of a piezoelectric substrate vibrating at nanometer displacement amplitudes ξ. Through a perturbation expansion, the governing equations of fluid motion are decomposed into those describing a first-order acoustic field and second-order acoustic streaming. Numerical solution of these equations and use of Fourier transforms allow the fundamental and harmonic frequencies of the surface deformation to be identified. For low excitation amplitudes ($\xi \sim 1$ nm), the frequency of the perturbed free surface is approximately equal to the SAW excitation frequency. However, as the amplitude increases ($\xi > 1$ nm), the dominant resonant frequency of the fluid free surface shifts to the low frequency range (f $\sim$ 1 MHz), suggesting that, in this regime, the free surface deformation is controlled by acoustic streaming. The numerical results for $\xi \sim 1$ nm qualitatively agree with experimental laser Doppler Vibrometry measurements.

12:01PM BQ.00008 Instability of Viscoelastic Thin Films, and Applications , SCOTT NORRIS, MICHAEL AZIZ, MICHAEL BRENNER, School of Engineering and Applied Sciences, Harvard University — An ion beam bombarding a solid surface has been long been known to produce an instability leading to a modulated surface (with ripples or dots); though the basic mechanisms for this instability remain under considerable debate. During our investigation of this problem, we have been led to a basic problem in thin film fluid mechanics: the instability of a viscoelastic thin film that is under compressive stress. This applies to the ion bombarded problem because there is evidence that the ion beam fluidizes a thin viscoelastic layer, and that this layer is then stressed by the ion beam. By varying the ratio of the shear modulus to the viscosity, we analyze this problem and connect the known limits of a stressed elastic solid film or a surface-tension driven lubrication flow. In particular, we identify the presence or absence of a surface instability as a function of these two parameters. We discuss the application of this model to the surface layer of an ion-sputtered target; and also discuss its potential application to the wrinkling instability of a growing biofilm attached to a substrate.

12:14PM BQ.00009 Electrohydrodynamic instabilities in thin bilayer liquid films , SCOTT ROBERTS, SATISH KUMAR, Department of Chemical Engineering and Materials Science, University of Minnesota, Minneapolis, Minnesota 55455 — When DC or AC electric fields are applied to a thin liquid film, the interface may become unstable and form a series of pillars. We examine how the presence of a second liquid interface influences pillar dynamics and morphologies. For perfect dielectric films, linear stability analysis of a lubrication-approximation-based model shows that the root mean square voltage governs the pillar behavior. For leaky dielectric films, Floquet theory is applied to carry out the linear stability analysis, and reveals that the accumulation of free charge at each interface depends on the conductivities in the adjoining phases and that high frequencies of the AC electric field may be used to control this accumulation at each interface independently. Nonlinear 1-D and 2-D simulations confirm the results of the linear stability analysis and reveal the final pillar morphology. The results presented here may of interest for the controlled creation of surface topographical features in applications such as patterned coatings and microelectronics.

12:27PM BQ.00010 Wetting dynamics of thin liquid films and drops under Marangoni and centrifugal forces[1] , SHOMEEK MUKHOPADHYAY, Chemistry Department, Columbia University, ROBERT BEHRINGER, Physics Department, Duke Univeristy — We present results from ongoing experimental studies on thin liquid drops and thin-films under the combined action of centrifugal forces due to rotation and radial Marangoni forces by using a temperature gradient. For thick rotating film in the absence of a temperature gradient, when an initially thick layer of fluid is spun to angular velocities where the classical Newtonian solution is negative, the fluid never dewets for the case of a completely wetting fluid, but leaves a microscopic uniform wet layer in the center. Similar experiments with a radially inward temperature gradient reveal the evolution of a radial height profile given by h(r) = A(t)r α, where A(t) decays logarithmically with time, and $\alpha = 0.8$. In the case where there is no rotation, small centrally placed drops show novel retraction behavior under a sufficiently strong temperature gradient. This work includes collaboration with Lou Kondic (NJIT), Nebojsa Murisic (UCLA) and Rich Mclaughlin (UNC-Chapel Hill).

[1]Work supported by the National Science Foundation under grant number DMS-02444498.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BR Instability: Jets & Wakes I 200F

10:30AM BR.00001 Stability and adjoint-based control of a confined impinging jet , JEAN-MARC CHOMAZ, LadHyX-CNRS, PHILIPPE MELIGA, LadHyX — We investigate numerically the dynamics of a laminar plane jet impinging on a flat plate in a channel. A global stability analysis is carried out and shows that, for a strong confinement, the two-dimensional steady flow is unstable to three-dimensional steady perturbations. We then use adjoint methods and sensitivity analyses to assess the efficiency of a 3-D harmonic or 2-D steady control to stabilize the leading 3-D global mode, by means of either bulk or wall forcing. This allows to identify flow regions of particular interest, and suggests different locations of the actuator depending on the control method. These concepts will be applied to two open-loop control strategies in which we introduce into the flow a small control device chosen as a cylinder or a flat-plate airfoil, modeled by the drag or lift force it exerts on the flow. A physical interpretation for the stabilizing effect of such control methods will be proposed, based on the competition between production and advection of disturbances.

10:43AM BR.00002 ABSTRACT WITHDRAWN –

10:56AM BR.00003 Shear Layer Instabilities in Low Density Transverse Jets[1]

, DANIEL GETSINGER, KEVIN CANZONIERI, CORY HENDRICKSON, OWEN SMITH, ANN KARAGOZIAN, UCLA Department of Mechanical and Aerospace Engineering — Shear layer instabilities associated with the gaseous, isodensity jet in crossflow have been explored in detail in recent experiments[2], indicating that the jet shear layer is globally unstable when the jet-to-crossflow velocity ratio, R, is less than 3.2 for a flush injected jet. Low density jets in quiescent surroundings are also known to become globally unstable for jet-to-ambient density ratios below approximately 0.6-0.7. It is thus of interest to explore the nature of changes in the character of shear layer instabilities for the low density jet in crossflow, with special focus on the influence of jet-to-crossflow momentum flux ratios at which instabilities are altered. A specially designed mixing device is utilized for exploration of helium and nitrogen jet mixtures. Calibration of the mixing device is accomplished using an acoustic waveguide capable of exploring alterations of standing wave frequencies with different gas mixtures. A range of flow conditions are explored, and alterations in the jet's spectral character suggesting transition to absolute instability are quantified.

[1]Supported by NSF grant CBET-0755104 and the NASA GSRP
[2]Megerian, et al., **JFM**, 593, pp. 93-129, 2007

11:09AM BR.00004 Closed- and Open-Loop Control of Isodensity Jets in Crossflow[1]

, CORY HENDRICKSON, DANIEL GETSINGER, JULIETT DAVITIAN, ROBERT M'CLOSKEY, ANN KARAGOZIAN, UCLA Department of Mechanical and Aerospace Engineering — Recent experiments have explored the behavior of a gaseous, isodensity jet in crossflow, focusing in part on the jet's response to controlled, strategic acoustic forcing[2]. While it is possible for strong sinusoidal jet excitation to overtake the natural shear layer instability when the flow is globally unstable (for jet-to-crossflow velocity ratios $R < 3.2$), square wave excitation of the jet fluid is observed to have a more profound effect on jet penetration and spread. Although creating precise square wave excitation for a globally unstable jet is challenging, open-loop control is observed to have some success. Yet at very small values of R, open-loop control is less capable of overcoming the instabilities and hence closed-loop control, whereby the forcing conditions continuously adjust to track a desired output square wave, must be used. The closed-loop controller, employed especially for $R < 1.25$, is observed to eliminate much of the distortion seen in the open-loop generated square wave, more closely matching the ideal square wave given bandwidth limitations of the actuation system, and providing robust control of the flow.

[1]Supported by NSF grant CBET-0755104 and the NASA GSRP.
[2]J. Davitian, PhD dissertation, UCLA, 2008

11:22AM BR.00005 Linear Stability Analysis of Round and Serrated Jets[1]

, KRISTJAN GUDMUNDSSON, TIM COLONIUS, California Institute of Technology — We investigate the velocity and pressure fluctuations of turbulent jets produced by round, and serrated nozzles. We model these fluctuations via the normal modes of the linearized equations and derive a generalized Rayleigh-equation for mean-flows composed of an arbitrary number of azimuthal harmonics, allowing fast solution of both the temporal and spatial stability problems. Using ensamble-averaged turbulent mean-flows from stereoscopic PIV data, we solve the generalized Rayleigh-equation and compare our predictions to instantaneous PIV measurements as well as near-field microphone measurements. Using the proper orthogonal decomposition to filter out uncorrelated fluctuations in data, we find good agreement between data and theory.

[1]Work supported by the AeroAcoustics Research Consortium (AARC).

11:35AM BR.00006 Experimental study of two opposing round jets

, HIND ALKANDRY, KOEN STEGEMAN, HIROSHI HIGUCHI, Syracuse University — The interaction between two opposing jets submerged in water is studied using particle image velocimetry. The research under present canonical configuration compliments an ongoing personal ventilation project conducted at Syracuse University. The study is carried out using two 9mm opposed round jets at two different angles separated by 25 diameters, one at zero degrees (directly opposing) and another at ten degrees. The Reynolds number based on the jet diameter is 9200, and a fully developed pipe flow exists at the exit. The mean velocity profile near the impact region shows a clear saddle point for both angles and radial jet emanating from that region. However, the instantaneous velocity field of the radial jet is highly unsteady. Three-dimensional characteristics, in particular at ten degrees, of the flow are examined closely.

11:48AM BR.00007 On the stability of a recirculation bubble and its application in compact combustion[1]

, MATT ANDERSON, PAUL STRYKOWSKI, University of Minnesota — A 2D channel flow expands asymmetrically via a sudden expansion splitter plate, the Reynolds number based on the channel height and mean velocity is 1.47×10^4. A recirculation bubble generated by a momenturm-driven counter flowing secondary stream located downstream of the sudden expansion is experimentally investigated by means of hot-wire anemometry and PIV. It is conjectured that the fluid field created is one of a separated region of locally absolutely unstable flow. This separated region is the result of both the partial stagnation of the main flow due to the spreading of the second counter-current flow as well as the entrainment of the secondary jet. The secondary stream augments the initial shear layer that has been created after the expansion and a separation bubble appears. This secondary jet creates a control mechanism for the fluid field. The low-velocity zone downstream of the expansion that has been created is necessary for flame anchoring, and the large turbulence levels recorded (total turbulence levels exceeding 100% of the inlet velocity) dramatically increase mixing and may lead to more efficient compact combustion in backward-facing step-combustors.

[1]The authors would like to acknowledge the generous support of the Office of Naval Research and the guidance that we have received from the technical monitor, G. D. Roy.

12:01PM BR.00008 Spectral analysis of nonlinear flows

, CLARENCE ROWLEY, Princeton University, IGOR MEZIC, UC Santa Barbara, SHERVIN BAGHERI, PHILIPP SCHLATTER, DAN HENNINGSON, KTH, Stockholm — We present a technique for describing the global behavior of complex, nonlinear flows, by decomposing the flow into modes determined from spectral analysis of the Koopman operator, an infinite-dimensional linear operator associated with the full nonlinear system. These modes, referred to as Koopman modes, are associated with a particular observable, and may be determined directly from data (either numerical or experimental) using a standard Arnoldi algorithm. They have an associated temporal frequency and growth rate and may be viewed as a nonlinear generalization of global eigenmodes of a linearized system. They provide an alternative to Proper Orthogonal Decomposition, and in the case of periodic data the Koopman modes reduce to a discrete temporal Fourier transform. We illustrate the method on an example of a jet in crossflow, and show that the method captures the dominant frequencies and elucidates the associated spatial structures.

12:14PM BR.00009 Turbulence and Internal Waves in a Stratified Jet , HIEU PHAM, SUTANU SARKAR, University of California, San Diego — Interaction between a stably stratified jet and internal gravity waves from an adjacent shear layer with mild stratification is investigated using 3D Direct Numerical Simulation. The streamwise velocity mimicks the Equatorial Undercurrents where a mixed shear layer situates on top of a strongly stratified jet. Despite the strong stratification, enhanced dissipation is observed in the jet. The evolution of the shear layer includes shear instability, Kelvin-Helmholtz rollers and subsequent breakdown to turbulence. Internal waves with wavelength larger than that of the rollers are found in and below the jet. The characteristics of the wave field follows linear theory. Analysis of the fluctuating energy budget indicates a balance mostly between the production and the transport for the wave field in the jet. The rate of change in the fluctuating kinetic energy in the wave field below the jet is balanced by the transport and the buoyancy flux. Hot fluid from the shear layer is entrained into the upper-flank of the jet, initiates turbulence and disrupts the internal wave field. The dissipation in the coherent patches of turbulence inside the stably stratified jet is strong, comparable to one inside the shear layer and up to three orders of magnitude stronger than that in the propagating wave field.

12:27PM BR.00010 Local concentration and velocity distribution for buoyancy driven mixing flows in long tubes at different tilt angles , J. ZNAIEN, F. MOISY, D. SALIN, J.-P. HULIN, FAST Laboratory, CNRS, Pierre et Marie-Curie Paris 6 and Paris-Sud 11 Universities (France), E.J. HINCH, DAMTP-CMS, Cambridge University (UK) — The buoyancy driven mixing of two fluids of different densities (Atwood numbers $10^{-3} \leq At \leq 10^{-2}$) interpenetrating each other has been studied at the local scale in a $20\mathrm{mm}$ diameter tube tilted at an angle $15^\circ \leq \theta \leq 60^\circ$ from vertical. The velocity and concentration maps are measured by means of PIV and LIF techniques in a vertical diametral plane. At large angles ($\theta = 45 - 60^\circ$) and low density contrast ($At = 10^{-3}$) the flow is laminar with three layers of different densities stabilized by transverse gravity. At high At ($4 \times 10^{-3} - 10^{-2}$) and low θ ($15^\circ - 30^\circ$), there is a turbulent shear mixing region with linear velocity and concentration transverse profiles in the middle of the tube section and with two channels of less mixed fluids at the top and bottom. The size of the channels increases at lower At and higher θ. The local viscous and turbulent momentum fluxes have been determined and their variation in the section and their dependence on θ will be discussed. At intermediate At and θ values, isolated or periodic turbulent bursts between which the flow returns to laminar are observed and both the velocity and concentration spatial correlations have been determined.

Sunday, November 22, 2009 10:30AM - 12:40PM –

Session BS Geophysical: Atmospheric II - Simulation 200G

10:30AM BS.00001 Direct Simulation of the Turbulent Ekman Layer: Evaluation of Closure Models , SCOTT WAGGY, STUART MARLATT, SEDAT BIRINGEN, University of Colorado at Boulder — A direct numerical simulation (DNS) at a Reynolds number of 400 was performed for the atmospheric boundary layer (ABL) using an Ekman layer approximation. Turbulence energy budgets show that these simulations are pertinent to the ABL. First-order closure models also compared well with the DNS results as the general distribution of the eddy diffusivity was captured correctly for both the neutral Ekman layer as well as the stably-capped Ekman-layer. This led to the conclusion that the vertical profile of the eddy diffusivity can be accurately represented by a cubic polynomial as is done in both first-order closure models.

10:43AM BS.00002 DNS of stably stratified open channel flow , OSCAR FLORES, JAMES RILEY, University of Washington — The surface layer of the atmospheric boundary layer under stable conditions (i.e., night time) is approximated here by a DNS of a stably stratified open channel, with buoyancy effects included using the Boussinesq approximation. The Reynolds number is $Re_\tau = u_* h/\nu = 560$, where u_* is the friction velocity and h is the height of the channel. Night time cooling is simulated by a negative heat flux at the ground, which imposes a stable density profile in the flow. The simulations are initialized with velocity fields obtained from a non-stratified case, and they are run until a quasi- steady state is achieved. As a result of the ground-cooling, the Reynolds stresses and the turbulent fluctuations decay in times of the order of L/u_*, where L is the Obukov length-scale. For relatively weak cooling, the turbulence survives, and the flow evolves towards a quasi-steady state. If the cooling is too strong, the flow becomes laminar. Our results indicate that parameter controlling that behavior is Lu_*/ν, rather than L/h as proposed before. Interestingly, for the turbulent cases, the flow in the near-wall region is very similar to the non-stratified case, except for the longest scales of the streamwise velocity. In the outer region the profiles of velocity fluctuations are also similar, but the turbulence structure and the energy balance are drastically changed by the stable stratification. Funded by ARO Grant No. W911NF-08-1-0155.

10:56AM BS.00003 ABSTRACT WITHDRAWN –

11:09AM BS.00004 Simulated statistics of polydisperse sedimenting inertial particles in a turbulent flow under experimental conditions[1] , LIAN-PING WANG, HOSSEIN PARISHANI, U. Delaware, BOGDAN ROSA, Institute of Meteorology and Water Management, Poland, COLIN BATESON, ALBERTO ALISEDA, U. Washington, WOJCIECH GRABOWSKI, NCAR — In recent years, point-particle based or hybrid direct numerical simulations (DNS) have increasingly been used to study pair statistics of inertial particles relevant to turbulent collision of cloud droplets. Equivalent experiment data are rare but are slowly becoming available. In this talk, we will discuss simulated statistics of sedimenting inertial particles under conditions similar to our parallel wind-tunnel experiment (to be reported here by Bateson et al.). The key parameters to be matched are flow Reynolds number, dissipation rate, particle Stokes number, and dimensionless settling velocity. A prescribed droplet size distribution will be used in the simulation to reproduce the polydisperse condition in the experiment. High-resolution DNS will be used to maximize the computational domain size. Single-particle and particle-pair statistics (e.g., fluctuation velocities, radial distribution function, relative velocity statistics) will be compared to the experimental data. Statistics obtained from lower dimensions will be linked to statistics in three dimensions.

[1]Work supported by NSF and NCAR.

11:22AM BS.00005 Reduced Order Estimation of the Atmospheric Boundary Layer Using POD-LSE , JONATHAN NAUGHTON, MANJINDER SINGH, University of Wyoming, EDWARD PATTON, PETER SULLIVAN, National Center for Atmospheric Research — The complimentary POD-LSE (proper orthogonal decomposition – linear stochastic estimation) technique has been used to develop a reduced order model from large eddy simulations (LES) of the atmospheric boundary layer. The technique allows modeling of the coherent turbulence that is inherent to real atmospheric flows, where older spectrum based methods fail. The usefulness of POD is the low number of modes required to capture the energy containing structure in the flow. The power of LSE lies in its ability to estimate the time-dependent flow field using sparsely distributed data points. For the daytime unstable atmospheric boundary layer studied here, approximately 90% of the total energy was captured by accurate estimation of less than 2% of the POD modes. Although the approach was developed using a Large Eddy Simulation as input, the approach provides the foundation for designing a field experiment. The POD modes would be determined from spatially-resolved measurements that lack time resolution, and the flow field dynamics would then be estimated using a few optimally placed sensors providing time-dependent information.

11:35AM BS.00006 LES of wind turbine wakes: Evaluation of turbine parameterizations , FERNANDO PORTE-AGEL, University of Minnesota, YU-TING WU, LEONARDO CHAMORRO — Large-eddy simulation (LES), coupled with a wind-turbine model, is used to investigate the characteristics of wind turbine wakes in turbulent boundary layers under different thermal stratification conditions. The subgrid-scale (SGS) stress and SGS heat flux are parameterized using scale-dependent Lagrangian dynamic models (Stoll and Porte-Agel, 2006). The turbine-induced lift and drag forces are parameterized using two models: an actuator disk model (ADM) that distributes the force loading on the rotor disk; and an actuator line model (ALM) that distributes the forces on lines that follow the position of the blades. Simulation results are compared to wind-tunnel measurements collected with hot-wire and cold-wire anemometry in the wake of a miniature 3-blade wind turbine at the St. Anthony Falls Laboratory atmospheric boundary layer wind tunnel. In general, the characteristics of the wakes simulated with the proposed LES framework are in good agreement with the measurements. The ALM is better able to capture vortical structures induced by the blades in the near-wake region. Our results also show that the scale-dependent Lagrangian dynamic SGS models are able to account, without tuning, for the effects of local shear and flow anisotropy on the distribution of the SGS model coefficients.

11:48AM BS.00007 Large Eddy Simulations of boundary layer flow over fractal trees[1] , JASON GRAHAM, Johns Hopkins University, CHARLES MENEVEAU, Mechanical Engineering and Center for Environmental and Applied Fluid Mechanics, Johns Hopkins University — A large-eddy simulation (LES) of flow over a canopy of fractal trees in the atmospheric boundary layer (ABL) is performed. The fractal trees provide complex boundary- turbulence interactions while maintaining tractable characteristics that can be systematically studied. LES are performed using the immersed boundary method following the implementation of Chester et al. (2007, J. Comp. Phys.). Simulations are performed for each fractal generation and the Reynods stresses and drag forces are computed. The LES results are used to analyze the impact of multiscale geometry on the Reynolds stress distribution and drag forces as a function of the generation number. This effort is also a feasiblity study for Renormalized Numerical Simulations (RNS) which is a methodology that allows for drag forces of unresovled generations to be computed by combining renormalizing techniques with the information from the resolved generations. Results are used to gain insight on scaling relationships between the drag forces and the generation number, and ultimately lead to better renormalization techniques for RNS.

[1]This research is supported by NSF (IGERT on Complex Systems), as well as NSF - ATM 0621396.

12:01PM BS.00008 Large Eddy Simulation of Persistent Contrails in Wind Shear and Atmospheric Turbulence , ALEXANDER NAIMAN, FRANK HAM, SANJIVA LELE, JORDAN WILKERSON, MARK JACOBSON, Stanford University — A study of contrail evolution was conducted using a three-dimensional Large Eddy Simulation (LES). The LES solves the incompressible Navier-Stokes equations with a Boussinesq approximation for buoyancy forces on an unstructured periodic grid. 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 microphysical model of ice deposition and sublimation. The simulation is initialized with the wake of a commercial jet superimposed on a decaying turbulence field. The ambient atmosphere is stable and has a supersaturated relative humidity with respect to ice. Grid resolution is adjusted during the simulation, allowing higher resolution of flow structures than previous studies. We present results of a parametric study in which ambient turbulence levels, vertical wind shear, and aircraft type were varied. We find that higher levels of turbulence and shear promote mixing of aircraft exhaust with supersaturated ambient air, resulting in faster growth of ice and wider dispersion of the exhaust plume. These results provide sensitivity data that improves understanding of the development of persistent contrails into contrail cirrus, a poorly characterized aspect of the climate impact of aviation.

12:14PM BS.00009 Large-eddy Simulation of Urban Boundary Layer Flow over Complex Topologies , BYUNG-GU KIM, CHANGHOON LEE, Yonsei University — Boundary layer flows over arrays of regularly distributed obstacles and a scaled real urban area in which various wind directions are considered were investigated by wind-tunnel experiment and large eddy simulation. Virtual boundary method is employed to represent the immersed complex geometries. Inflow conditions are generated such that given profiles of mean wind and turbulence properties such as integral length or time scales are matched. Constant Smagorinsky subgrid-scale model is used. Surface flow parameters such as friction velocity (u_*), roughness length (z_0) and displacement thickness (d) were evaluated by changing wind direction. It was found that the parameters for the arrays composed of slender rectangular cylinders whose characteristics are similar to a real urban area than cube arrays are highly sensitive to the wind direction. Many previous works that have been focused on cube arrays would differ from the real urban area in the characteristics of flow field. Large-eddy simulations were extended to the region of Daejeon city, Korea to study the province's characteristics. Velocity profile along the street canyon with which a wind direction aligned were closed to a linear profile rather than a constant or exponential distribution. Detailed wind field characteristics above and below the canopy will be presented in the meeting.

12:27PM BS.00010 A dynamic subgrid-scale parameterization of the effective wall stress in atmospheric boundary layer flows over multiscale, fractal-like surfaces[1] , WILLIAM ANDERSON, Johns Hopkins University, CHARLES MENEVEAU, Mechanical Engineering and Center for Environmental and Applied Fluid Mechanics, Johns Hopkins University — A dynamic subgrid-scale parameterization for hydrodynamic surface roughness is developed for large-eddy simulation (LES) of atmospheric boundary layer (ABL) flow over multiscale, fractal-like surfaces. The model is tested on surfaces generated through superposition of random-phase Fourier modes with prescribed, power-law surface-height spectra. Surfaces are generated at fine-grained resolution and subsequently spatially filtered to various affordable LES resolutions. The lower boundary condition is prescribed based on the logarithmic law of the wall, where the unresolved roughness from the fractal surface is modeled as the product of local root- mean-square (RMS) of the unresolved surface height and an unknown dimensionless model coefficient. This coefficient is evaluated dynamically by comparing the plane- average wall-stress at two resolutions (grid- and test-filter scale, Germano et al., 1991). The results show that the method yields convergent results and correct trends. Limitations and further challenges are highlighted.

[1]Supported by NSF (EAR-0609690).

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BT Vortex Dynamics and Vortex Flows II 200H

10:30AM BT.00001 Deriving Kelvin's argument through the principle of "virtual vortex work" , P. LUZZATTO-FEGIZ, C.H.K. WILLIAMSON, Cornell University — In 1875, Lord Kelvin proposed an energy argument to define equilibrium and stability in fluid flow. Kelvin stated that steady flows would realize a stationary point of the energy, for given vorticity and impulse. Intriguingly, Kelvin proposed this idea without proof; analytical confirmation was presented a century later by Benjamin (1975). Unfortunately, to this date, we have no indication as to how Kelvin's argument may be derived from a fundamental physical principle. Indeed, the path that led Kelvin to his statement remains unknown. A derivation based on a fundamental principle may enable generalizations to novel applications, which cannot be investigated by the current formulation of Kelvin's argument. In this presentation, we employ the fundamental principle of virtual work, and show that the requirement for a system to be stationary in a *moving* frame leads naturally to a constraint on the impulse. In the context of fluid flow, formulating the principle leads us to introduce the concept of *virtual vortex work*. We show this to be equivalent to Kelvin's argument. We exploit our derivation to devise generalizations of Kelvin's argument for a variety of fluid flows, including vortical flows, gravity waves and compressible flows. This, in turn, allows us to instantly deduce stability properties in 2D and 3D through an "imperfect-velocity-impulse" approach (which takes the form of "IVI" diagrams) recently proposed by the authors.

10:43AM BT.00002 Time resolved measurements of vortex-induced-vibration of a tethered sphere , RENE VAN HOUT, ALEXANDER KRAKOVICH, ODED GOTTLIEB, Technion-Israel Institute of Technology — Time resolved, high-speed PIV measurements were performed to study the spatio-temporal dynamics of a tethered, stainless steel sphere ($D = 5/16$", m^*=7.86, L^*=2.90), mounted in a water channel and exposed to uniform, free stream velocities up to 0.64 m/s. To study the coupled interaction between vortex and sphere dynamics, we performed measurements in a horizontal plane, intersecting the sphere's center. Below the threshold velocity for which vortex induced vibration occurs, wake dynamics are those of a stationary sphere. As the free stream velocity increases, modes of periodic and non-stationary intermittent sphere dynamics are observed in the plane transverse to the flow. We simultaneously track the sphere and vortex centers, the latter through maximum values of the swirling strength. Vortex tracking starts at the sphere interface through the separation point and away from the sphere. Also, time dependent turbulent stresses, terms of the TKE production are presented. This information together with the sphere's motion itself reveals clues to the intricate, coupled flow-structure interaction.

10:56AM BT.00003 A mathematical model of a "2P mode" vortex wake , ALIREZA SALMANZADEH, PhD Candidate at ESM Department, Virginia Tech, MARK STREMLER, Assistant Prof. of ESM Department, Virginia Tech — The standard von Karman vortex street, also known as the "2S" mode, is the most common vortex configuration to appear in the wake of a bluff body. The next most common configuration is the "2P" mode, in which two pairs of vortices are shed per cycle. We consider a simple model of the "2P" mode consisting of a singly-periodic Hamiltonian system of four point vortices with identical strength magnitudes and zero net strength. An imposed spatial symmetry results in integrable dynamics that depend only on the relative vortex positions. Comparison of our model with a recent experimental result (Schnipper, Andersen, and Bohr, JFM 2009) suggests that this model approach can be used to characterize the experimental vortex motion and estimate the experimental vortex strengths.

11:09AM BT.00004 Vorticity Measurements in the Wake of an Inclined Prolate Spheroid , KURT KELLER, ALAN BRANT, KEN KALUMUCK, JHU/APL, CHARLES SCHEMM, STEVE SCORPIO — The generation and evolution of axial vorticity in the wake of an inclined 6:1 prolate spheroid is studied experimentally, with comparison to Computational Fluid Dynamics (CFD) results. 2D Particle Image Velocimetry (PIV) measurements were obtained in planes normal to the flow at several stations along the body and at downstream distances up to one body length, at angles of attack of 5, 10, and 20 degrees and body Reynolds numbers (Re_L=UL/ν) of $\{13.7, 27.3, 45.6\} \times 10^4$. As an extension of previous numerical and experimental studies on the vortex roll-up on the body of a 6:1 Prolate Spheroid [for example, Fu *et al* (1994), Tsai and Whitney (1999)], this study is focused on characterizing the downstream vorticity distribution as a function of the angle of attack and body Reynolds number. Long time average measurements of the circulation, core size, and core location are presented as a function of the angle of attack and the free stream velocity. In addition, measurements of turbulence characteristics of the wake are presented. Vortex migration velocities are found to be less than those estimated from inviscid vortex dipole theory. Experimental results for the 10-degree case are compared. Reynolds Average Navier-Stokes (RANS) CFD calculations show significant differences in the vorticity distribution near the stern, but with good agreement at one body length downstream.

11:22AM BT.00005 Organization of Cylinder Wake Using a Splitter-Plate Active Flow Control , PAVLOS VLACHOS, CHRIS WEILAND, Virginia Tech — It is well-known that a splitter plate in the wake of a circular cylinder prohibits the formation of the classic von Karman vortex street. Here we present an experimental study which shows the near wake can be manipulated using flow control to restore the vortex shedding in the presence of a splitter plate. Three splitter plate locations along with three cylinder diameters were analyzed using spectral analysis and proper orthogonal decomposition of time resolved digital particle image velocimetry (TRDPIV) data. As an example, in one case the splitter plate was placed 1.9 diameters downstream of the cylinder and spectral measurements of the TRDPIV results indicated its presence decreased the Strouhal number from 0.19 to 0.12 as anticipated. When activated the flow control restored the wake to a Strouhal number of 0.19 and a 2P vortex shedding mode was clearly visible. The data suggests that the jet excited the circular cylinder shear layers causing instability, roll up, and subsequent vortex shedding.

11:35AM BT.00006 Experimental Study of Synchronization and Phase Dynamics in Flapping Wing Propulsion , CYNDEE FINKEL, KARL VON ELLENRIEDER, Florida Atlantic University — Experiments are conducted on a two dimensional heaving airfoil to determine whether or not natural flight can be modeled as a limit cycle process as well as the degree to which the wing motion, any vortices shed from upstream bodies and the fluid force response act as dynamically coupled oscillators. The heaving airfoil mechanism is constructed to permit the experimental simulation of a freely flying system in which the forward speed of the system is determined by the Strouhal number and reduced frequency of the motion. Also examined is how the structure of the flow behind a freely flying system differs from that of a strongly forced system, where the position of a flapping airfoil mechanism is fixed and the forward velocity is imposed.

11:48AM BT.00007 Vortex rings behind an oscillating sphere , VIATCHESLAV MELESHKO, Kiev National University, ALEXANDRE GOURJII, Institute of Hydromechanics, NAS Ukraine — The talk presents the results of an investigation of the formation, stability and control of localized vortex ring structures due to a periodic motion of a sphere in an inviscid incompressible fluid. The low order model based upon Dyson's vortex rings is employed. The intensity of generated vortex rings is estimated by the total vorticity, which takes place in a viscous boundary layer on the sphere. Numerical results of simulations of the transport processes in the near-wall zone, the determination of regions of low and high domains of passive admixtures are discussed. Finally, we compare numerical and analytical solutions with the results of analogue laboratory experiments for helium II.

12:01PM BT.00008 Dynamics of collision of a vortex ring and a planar surface[1] , MICHAEL MCERLEAN, MICHAEL KRANE, ARNOLD FONTAINE, ARL Penn State — The dynamics of the impact between a vortex ring and a planar surface orientated perpendicular to the direction of travel are presented. High Reynolds number vortex rings are injected into a quiescent tank of water using a piston-cylinder generator before colliding with a target at a long distance. Both the pressure at the stagnation point on the surface and the force imparted to the target by the ring impact are measured directly. The changes in both are related to the ring motion and deformation captured by high speed digital video, and DPIV measurements. These relations are used to develop a scaling law relation between impact force and vortex ring circulation, speed, and size.

[1]Acknowledge support of NAVSEA and NIH.

12:14PM BT.00009 Strain dynamics for vortex ring mixing process[1] , YANN BOUREMEL, MICHAEL YIANNESKIS, ANDREA DUCCI, Kings College London — Simultaneous PIV-PLIF measurements were carried out to investigate the mixing occurring in a laminar vortex ring flow during the formation stage (Re=357-1072). In the first part of the work a control volume analysis was used to determine the variation in time of the scalar concentration mean, variance, and probability density function. In the second part the advection-diffusion differential equations of the scalar, ξ, and of its energy, 0.5 ξ^2, were studied in depth to gain insight into the effect of the strain rate tensor, $\mathbf{S}$, on the local scalar concentration for increasing *Re*. The measurements were obtained with a high spatial resolution (12 μm for the PLIF) in order to resolve the scalar dissipative scales. Reliable estimates of the scalar dissipation rate ($\nabla\xi \cdot \nabla\xi$), and of the symmetric contraction term ($\nabla\xi \cdot \mathbf{S} \cdot \nabla\xi$), shown in equation 1, were obtained. $\nabla\xi \cdot \mathbf{S} \cdot \nabla\xi$ accounts for the reduction of scalar dissipation due to the straining component directed as the local scalar gradient (see Southerland et al.[2]) Equation 1: $\left(\frac{\partial}{\partial t} + \vec{u}.\nabla + \frac{1}{ReSc}\nabla^2\right) \frac{1}{2}\left(\nabla\xi.\nabla\xi\right) = -\left(\nabla\xi.S.\nabla\xi\right) - \frac{1}{ReSc}\nabla(\nabla\xi) : \nabla(\nabla\xi)$

[1]Grant from EPSRC (EP/D032539/1).

[2]Southerland K B., Porter III J. R., Dahm, W. J. A., Buch K. A., An experimental study of the molecular mixing process in an axisymmetric laminar vortex ring, Phys. Fluids A 3 (5), May 1991

12:27PM BT.00010 Experimental Investigation of Vortex Ring Interaction with Inclined Surfaces , LAUREN D. COUCH, PAUL S. KRUEGER, SMU — A number of experimental and numerical studies have described the collision of a laminar vortex ring with an inclined surface and noted similarities with hairpin vortices found in turbulent boundary layers. However, the dependence of the observed flow on the vortex ring properties and angles of collision have largely been neglected. In the present investigation, vortex ring interactions with an inclined plate were studied experimentally to determine the effects of plate angle on the flow evolution and draw comparisons with coherent structures in turbulent boundary layers. Vortex rings were generated using a mechanical piston-cylinder vortex ring generator at jet Reynolds numbers ranging from 1000 to 3000 and stroke length-to-piston diameter ratios from 0.5 to 2. The plate angle relative to the initial axis of the vortex ring ranged from 3 to 70 degrees. Flow observations were made using planar laser induced fluorescence, 2D digital particle image velocimetry (DPIV), and 3D defocusing DPIV (DDPIV). Results show deformation and stretching of the vortex ring into a loop-like vortex and the generation of secondary vorticity at the surface of the plate.

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BU Granular II: Jamming II 200I

10:30AM BU.00001 Shearing and compression of elliptical particles[1] , SOMAYEH FARHADI, ROBERT BEHRINGER, Duke University — We have performed 2D biaxial shearing and compression experiments for elliptical photoelastic particles in order to understand the effect of particle shape on microscopic and macroscopic properties of a granular system. The shearing experiment was conducted via a series of small forward and reverse steps using pure shear. We study the evolution of particle orientations and the average number of contacts following each step of shear or compression. Using photoelastic particles enables us to visualize the stress state of the system at the particle scale level. The ongoing analysis addresses the statistical properties of jammed state, including jamming that is reached through compression or through shear.

[1]Work of RPB supported by NSF grant DMR0906908, ARO grant W911NF-07-1-0131-00, and BSF grant 2004391.

10:43AM BU.00002 Scaling of boundary stresses in a Couette flow of granular mixtures , BEREKET YOHANNES, KIMBERLY HILL, University of Minnesota — Most constitutive laws for dense granular flows that predict stresses based on particle size and shear rates are developed based on experiments and simulations of uniformly sized spherical particle. These types of constitutive laws have not yet been formulated for granular mixtures. In order to investigate the applicability of certain constitutive laws for dense granular mixtures, we study the boundary stresses due to a Couette flow of binary mixtures of different sized particles using a 3D discrete element method (DEM). For a given mixture, as in a uniform particle size distribution, the stresses scale with the square of the shear rate. However, the stress appears to have a more complicated dependence on both the relative sizes of the particles in the mixture and the relative concentration of the different species. In these simulations the coordination number is found to be a better quantitative parameter than average size to describe the stresses. We present the relationship between boundary stresses, particle size distribution, and coordination number.

10:56AM BU.00003 Scaling of boundary stresses in granular mixtures: free surface gravity-driven flows , KIMBERLY HILL, BEREKET YOHANNES, St. Anthony Falls Laboratory, Civil Engineering, University of Minnesota, WILLIAM DIETRICH, LESLIE HSU, Earth and Planetary Science, University of California - Berkeley — The ability to predict the stresses particulate mixtures exert on their boundaries is important for many critical natural and industrial applications. For example, predicting boundary stresses due to bouldery debris flows is important for understanding landscape morphology and associated hazard mitigation. Typically, models developed to predict internal and external stresses in these flows contain reference to a particle size and sometimes a maximum packing fraction, neither of which in known for a granular mixture. We find that, while for a mixture sheared in a Couette cell substitutions may be found for both terms, the situation is somewhat more complicated in a free surface flow relevant to debris flows. In the latter case the particle size distribution and the phase of flow is non-uniform in the direction of flow, and both neeed to be considered for determining the local and global stresses in the system.

11:09AM BU.00004 Shapes of elastic cylindrical tubes filled with liquids and granular media[1] , G. JULIANA GUTIERREZ, ABRAHAM MEDINA, ESIME Azcapotzalco, IPN — In this work we discuss theoretically and experimentally the deformation equilibrium shapes of the elastic walls of a vertical tube when it was filled with water and sand. The use of elastic soft walls has been motivated to note, through the deformation of the walls, the role of the pressures induced by each material. By using the linear elastic theory it is possible to estimate analytically the shapes of the deformed walls induced by the respective hydrostatic pressures of water and sand.

[1]Authors acknowledge SIP from IPN by the support of this study.

11:22AM BU.00005 Enhanced penetration forces with simultaneous granular intruders[1] , PAUL UMBANHOWAR, Northwestern University, LIONEL LONDON, YANG DING, DANIEL GOLDMAN, Georgia Institute of Technology — To better understand how the geometry and actuation of biological and mechanical feet affect locomotor performance on flowable ground, we examine the constant velocity insertion into granular media of two horizontal, parallel rods as a function of rod separation s. Our experiments and simulations show that while the force F required to maintain a constant velocity increases linearly with the penetration depth d in all cases, the slope F/d is a non-trivial function of rod separation. As s is increased from zero, F/d initially increases, reaches a maximum value at ≈ 2 grain diameters, and then slowly decreases to twice the value of F/d for a single rod at large separation. Examining force correlations and flow intermittency between the rods, we show that a model of cooperative arching and jamming explains the salient features of our results.

[1]This work was supported by the Burroughs Wellcome Fund.

11:35AM BU.00006 The design, testing, and performance of RoboClam, a robot inspired by the burrowing mechanisms of Atlantic razor clams (Ensis directus) , AMOS WINTER, ANETTE HOSOI, ALEXANDER SLOCUM, MIT — In this work we present the design, testing, and performance of RoboClam, a robot that mimics digging methods employed by the Atlantic razor clam (Ensis directus). Ensis is one of nature's most adept burrowing organisms, able to dig to 70cm at nearly 1cm/s using only 0.21J/cm. We have found that Ensis reduces burrowing drag by using motions of its shell to fluidize a thin layer of substrate around its body. Although these shell motions have an energetic cost, moving through fluidized rather than packed soil results in dramatically lower overall energy consumption. RoboClam was constructed to understand the limits of razor clam-inspired burrowing, how the relevant parameters scale for different environments and conditions, and how this understanding can be transferred into engineering applications. Through experimental data gathered in idealized granular, as well as real ocean substrates, we show that RoboClam exploits localized fluidization to attain nearly the same burrowing energy savings as Ensis.

11:48AM BU.00007 Measuring the phase diagram of granular media , J-F. METAYER, MPI for Dynamics and Self-Organization, D.J. SUNTRUP III, C. RADIN, H.L. SWINNEY, University of Texas, M. SCHROETER, MPI for Dynamics and Self-Organization — The jamming transition in granular media has been the subject of several studies in the last years. However, an experimental observation of phase diagram of granular media in three dimensions is still lacking. The goal of this study is to obtain such a diagram as a function of the shear stress, the packing fraction, ϕ, and the pressure in a granular bed. Shear stress is obtained by measuring the force, F, needed to pull-up a paddle immersed in a granular bed as a function of its packing fraction and the depth of immersion. We find that the value of this force is strongly depending the packing fraction: for low packing fraction ($\phi < 0.59$) F is constant while it increases strongly with ϕ for higher packing fraction.

12:01PM BU.00008 Stick-slip transition at the granular critical state , NICK GRAVISH, Georgia Tech, PAUL UMBANHOWAR, DANIEL I. GOLDMAN — We study the force on a flat plate (3.8 cm width, 7.0 cm depth) dragged at constant velocity v through the surface of a granular medium (250 μm glass beads) as a function of volume fraction $0.57 < \phi < 0.63$. The dynamics of the drag force F_d are sensitive to ϕ: we find a sharp transition in the form of F_d at a critical volume fraction $\phi_c = 0.605$. For $\phi < \phi_c$, F_d increases with time and saturates, while for $\phi > \phi_c$ F_d exhibits an initial peak followed by periodic oscillations at frequency f about a constant mean. The standard deviation in force (a measure of the fluctuations) shows a sharp transition at ϕ_c. The force oscillations suggest that the granular media periodically jams and flows as the plate is horizontally translated. Examining the bed surface we observe a spatially periodic scalloped feature of length λ which is equal to v/f, independent of v, and increases linearly with ϕ for $\phi > \phi_c$. By measuring the displaced volume after the drag ΔV, we observe a transition from media compaction ($\Delta V < 0$) for $\phi < \phi_c$ to dilation ($\Delta V > 0$) for $\phi > \phi_c$.

12:14PM BU.00009 Velocity field within a *stagnant zone* in a granular flow , ENRIQUE SANDOVAL, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — It is known that in granular flows stagnant zones may appear. It has been usually thought that these regions remain immobile; however creep-like motion has been observed. The objective of this study is to measure the velocity field within the stagnant zone. Are the flow properties of such zone determined by the material (particle size, maximum solid fraction, roughness) or the flow (velocity)? We hope to answer this question. A gravity-driven granular flow was produced with a vertical channel filled with glass beads. The mean speed was controlled with discharge hopper. A solid plate is positioned in the center of the chute; its interaction with the flow produces a stagnant region in its upstream side. Using an image correlation technique (normally used in the PIV method), velocity profiles were measured for different angles, mean discharge velocities and plate sizes. For the case in which plate is perpendicular to flow, the velocity was found to decay exponentially within stagnant region. The exponential decay ratio is proportional to the distance from the plate. In this talk the nature of this behavior and its implications for granular media rheology will be discussed.

12:27PM BU.00010 Column Collapse of Rod-like Granular Materials[1] , MELISSA TREPANIER, SCOTT FRANKLIN, Rochester Institute of Technology — We study the collapse of piles of rod-like granular materials, in particular how the particle aspect ratio (length/width) and coefficient of friction affect the runoff. Rod particles can maintain the shape of their container, something round particles cannot, and we find transitional pile heights that determine the onset of collapse. For low aspect ratios, pile heights of less than a particle length do not collapse, implying that vertically oriented rods are anchoring the pile and providing stability. There is a broad transition range of pile heights in which the probability of collapse grows linearly from 0 to 1. The scaling of the runoff distance in and above this region is independent of aspect ratio and friction, depending only on the initial pile geometry. This work could have significant implications for construction of stable structures and understanding avalanches of needle-like snow crystals (hoar).

[1]NSF DMR #0706353

Sunday, November 22, 2009 10:30AM - 12:40PM –
Session BV Flight II 205A-D

10:30AM BV.00001 Flow Field of Flexible Flapping Wings , ERIK SÄLLSTRÖM, LAWRENCE UKEILEY, University of Florida — The flow field around several flexible flapping Zimmerman planform wings of aspect ratio 7.65 and a semispan of 75 mm is investigated using particle image velocimetry (PIV) in a quiescent environment. The wings are made from carbon fiber skeletons and covered with a thin layer of Capran. The skeletons consist of reinforced leading edges and chordwise battens in an attempt to decouple chordwise and spanwise flexibility as much as possible. The flow field from several phases throughout the flapping cycle will be presented. These flow fields consist of the phase averaged velocities in multiple PIV planes. These planes will include both those orientated in the streamwise and spanwise directions to build up a three dimensional representation of the flow in the vicinity of the wing and calculate the resultant vorticity field. The vortical features of these flow fields will be identified and discussed through the use of vortex identification methods. The discussion of the flow measurements will be coupled with force measurements and wing deflection data for a detailed view of mechanisms related to flapping flight and study how the formation of vorticity relates to the generation of aerodynamic forces.

10:43AM BV.00002 Tumbling dynamics of flexible wings , DANIEL TAM, JOHN BUSH, MIT, MICHAEL ROBITAILLE, ARSHAD KUDROLLI, Clark University — We are broadly interested in elucidating the role of flexibility in passive flight. In particular, we examine the role of bending on the flight of autorotating winged seedpods through an experimental investigation of tumbling rectangular paper strips freely falling in air. Our results suggest the existence of a critical length above which the wing bends. We develop a theoretical model that demonstrates that this buckling is prompted by inertial forces associated with the tumbling motion, and yields a buckling criterion consistent with that observed. We further develop a reduced model for the flight dynamics of flexible tumbling wings, that illustrates the effect of aeroelastic coupling on flight characteristics and explains experimentally observed variations in the wing's falling speed and range. Other modes of flexible passive flight are discussed as well as biological implications for the dispersal of seed pods.

10:56AM BV.00003 An implemental formulation of Newton dynamics of free insect flight , SHENG XU, Southern Methodist University — A free-flying insect flies and maneuvers by coupling aerodynamics and Newton dynamics. In this talk, I will present a formulation of Newton dynamics for this coupling. This formulation is clear, concise, and simple for implementation. It passes a few basic tests.

11:09AM BV.00004 Aerodynamics of a single-degree-of-freedom toy ornithopter , RAMIRO CHAVEZ ALARCON, B.J. BALAKUMAR, JAMES J. ALLEN, New Mexico State University — The flow field around a flight-worthy toy ornithopter is investigated using PIV diagnostics in combination with load cells to understand the aerodynamics during nominally steady flight and turning. Phase-locked measurements of the wake and inflow are performed using an automated PIV system around the flapping wings of the ornithopter with the ornithopter fixed to a load-cell inside a 1.3m x 1.2m wind tunnel test section. The mildly oscillating free flight of the ornithopter is compared to the wake measurements to understand the causes of the unsteadiness. Further, the modulation of the wake that causes the turning motion of the ornithopter is explained using the wake structure measurements.

11:22AM BV.00005 Flapping-wing mechanical butterfly on a wheel[1] , RAMIRO GODOY-DIANA, BENJAMIN THIRIA, DANIEL PRADAL, PMMH UMR7636 CNRS; ESPCI; PARIS 6; PARIS 7 — We examine the propulsive performance of a flapping-wing device turning on a "merry-go-round" type base. The two-wing flapper is attached to a mast that is ball-bearing mounted to a central shaft in such a way that the thrust force produced by the wings makes the flapper turn around this shaft. The oscillating lift force produced by the flapping wings is aligned with the mast to avoid vibration of the system. A turning contact allows to power the motor that drives the wings. We measure power consumption and cruising speed as a function of flapping frequency and amplitude as well as wing flexibility. The design of the wings permits to change independently their flexibility in the span-wise and chord-wise directions and PIV measurements in various planes let us examine the vorticity field around the device. A complete study of the effect of wing flexibility on the propulsive performance of the system will be presented at the conference.

[1]Supported by the French National Research Agency through project ANR-08-BLAN-0099.

11:35AM BV.00006 Similarities and differences in the wake structure generated by different species of bats[1] , TATJANA HUBEL, NICKOLAY HRISTOV, SHARON SWARTZ, KENNETH BREUER, Brown University — Flight kinematics and morphology differ greatly between the approx. 1200 bat species and the goal of our project is to understand how these differences affect the flight mechanisms, the generation of aerodynamic forces, and the resultant wake structures. Multiple individuals of three diverse species of bat were flown in the wind tunnel. The three species have different morphology, wing aspect ratio and wing loading, and exhibit different flight behaviors appropriate to their different ecologies. Particle Image Velocimetry in the cross-stream (Trefftz) plane acquired at 200 Hz was used to map the time-resolved wake velocities behind the bat, while three synchronized high-speed cameras monitored the wing motion. The measurements were taken at several flight speeds. Early predictions based on kinematic measurements suggest the development of discrete vortex rings as well as the generation of negative circulation in the wing tip area during the end of the downstroke. These hypotheses are tested and discussed, and the results show distinct differences between the species and as functions of flight speed.

[1]Supported by AFOSR and NSF.

11:48AM BV.00007 Propulsive performance of oscillating batoid-inspired fins , DANIEL QUINN, University of Virginia, DAPHNE REIN-WESTON, PETER DEWEY, MELISSA GREEN, ALEXANDER SMITS, Princeton University — Thrust producing ray-like pectoral fins were actuated to drive a low friction carriage through a stationary tow tank. A DC servo motor powered a gear train that produced a traveling wave motion along the chord of the fin. The amplitude of the traveling wave increased linearly along the span from root to tip. A digital optical encoder attached to the carriage tracked the position and velocity as it was propelled through the water by the oscillating fin. Velocity profiles were acquired from trials using different planforms inspired by members of the eagle ray family, as well as an idealized elliptical fin. Traveling wave frequency and wavelength were varied to investigate the propulsive performance of different gaits. Preliminary flow visualization was also performed to describe the structure of the wakes generated by the various planform geometries and locomotory gaits.

12:01PM BV.00008 The fluid dynamics of mayfly naiads[1] , K. ABDELAZIZ, E. BALARAS, K. KIGER, University of Maryland — The present work is focused on the study of mayfly naiads (nymphs) as a possible biological model for the efficient generation of external circulation currents in low to intermediate Reynolds number flows. Our primary objective is to validate a series of high-fidelity simulations we conducted by comparisons to experimental results obtained at an earlier stage of the project. For this purpose a realistic 3D model of the mayfly is constructed. It includes the abdomen, thorax, head, and six pairs of gills. Each gill is represented by two, zero-thickness rigid plates, which are hinged to at the location of the primary flexion line. The kinematics for all gills are prescribed and derived from the experiments. In particular, the trajectory of a set of points recorded in the experiments is translated into a sequence of Euler angles, which are then fed to a three-level kinematic chain enabling the derivation of all rigid body kinematics in the inertial frame. A Navier-Stokes solver in Cartesian coordinates is used and boundary conditions on the complex moving bodies are imposed with an embedded-boundary, direct forcing approach. The phase averaged numerical results are in good agreement with the corresponding particle image velocimetry (PIV) data from the experiment. The effects of gill kinematics and Reynolds number will also be discussed.

[1]Research suported by the National Science Foundation.

12:14PM BV.00009 The Rufous Hummingbird in hovering flight – full-body 3D immersed boundary simulation , PAULO FERREIRA DE SOUSA, HAOXIANG LUO, Vanderbilt University, HUMBERTO BOCANEGRA EVANS, New Mexico State University — Hummingbirds are an interesting case study for the development of micro-air vehicles since they combine the high flight stability of insects with the low metabolic power per unit of body mass of bats, during hovering flight. In this study, simulations of a full-body hummingbird in hovering flight were performed at a Reynolds number around 3600. The simulations employ a versatile sharp-interface immersed boundary method recently enhanced at our lab that can treat thin membranes and solid bodies alike. Implemented on a Cartesian mesh, the numerical method allows us to capture the vortex dynamics of the wake accurately and efficiently. The whole-body simulation will allow us to clearly identify the three general patterns of flow velocity around the body of the hummingbird referred in Altshuler et al. (Exp Fluids 46 (5), 2009). One focus of the current study is to understand the interaction between the wakes of the two wings at the end of the upstroke, and how the tail actively defects the flow to contribute to pitch stability. Another focus of the study will be to identify the pair of unconnected loops underneath each wing.

12:27PM BV.00010 PIV-based study of the gliding osprey aerodynamics in a wind tunnel , ROI GURKA, Ben-Gurion U., ALEX LIBERZON, Tel-Aviv U., GREGORY KOPP, ADAM KIRCHHEFER, U. of Western Ontario, DANIEL WEIHS, Technion — The hunting flight of an osprey consists of periods where the bird glides while foraging for prey. High quality measurements of aerodynamics in this flight mode are needed in order to estimate the daily energy expenditure of the bird accurately. An experimental study of an osprey model in a wind tunnel (BLWTL, UWO) was performed in order to characterize the aerodynamic forces using particle image velocimetry (PIV). The model was a stuffed osprey with mechanical joints allowing control of the the wing (angle of attack, tilt) and tail orientation. Two-dimensional velocity realizations in the streamwise-normal plane were obtained simultaneously in the two fields of view: above the wing and in the wake of the wing. Mean and turbulent flow characteristics are presented as function of angle of attack based on measurements taken at 4 different angles of attack at three different locations over the wingspan. The main outcome is the accurate estimate of the drag from the measurements of momentum thickness in the turbulent boundary layer of the osprey wing. Moreover, the gradient of the momentum thickness method was applied to identify the separation point in the boundary layer. This estimate has been compared to the total drag calculated from measurements in the wake of the wing and with a theoretical prediction.

Sunday, November 22, 2009 10:30AM - 12:40PM –

Session BW Mini-Symposium on Fluid Dynamics of Sports 208A-D

10:30AM BW.00001 Modeling golf ball fluid mechanics - challenges and opportunities , KYLE SQUIRES[1], Arizona State University — Numerical simulation presents a powerful tool for understanding the fundamental fluid mechanics that influence golf ball aerodynamics, as well as providing an approach for ultimately analyzing and designing golf balls for manufacture. Robust and accurate simulation strategies are central to providing a means to screen designs prior to costly prototyping and field measurement. Results from a hierarchy of simulation strategies applied to the flow around golf balls will be presented, ranging from Reynolds-averaged Navier-Stokes (RANS) computations to Direct Numerical Simulation (DNS). RANS methods, while leading to computationally efficient approaches, are challenged to represent using ad hoc turbulence models the subtle effects induced by surface dimpling. DNS on the other hand, offers a first-principles approach that enables detailed examination of mechanisms though carries a significant computational cost. Predictions from both techniques are contrasted; opportunities for advancing each technique are identified.

[1]In collaboration with: Elias Balaras, University of Maryland

10:56AM BW.00002 Experimental measurement of the aerodynamic properties of golf balls , ALEXANDER SMITS[1], Princeton University — Accurate measurements of the lift, drag and spin rate decay characteristics of golf balls are necessary to predict the golf ball trajectory and its point of impact. Three principal methods are used to determine these characteristics: (1) wind tunnel testing using a lift and drag balance; (2) outdoor testing using a tracking camera, whereby the trajectory and spin of the golf ball is monitored during flight, and (3) indoor testing, whereby the trajectory is broken down into various Reynolds number regimes and each regime is tested by launching the golf ball over a heavily-instrumented short indoor range. Here, we discuss the relative merits of each testing technique, and demonstrate how the output of the tests is used in a trajectory program to simulate the flight of the ball.

[1]In collaboration with: Sean Bailey, Princeton University

11:22AM BW.00003 Computational Modeling and Analysis of the Fluid Dynamics of Competitive Swimming , RAJAT MITTAL, Mechanical Engineering, Johns Hopkins University — In order to swim efficiently and/or fast, a swimmer needs to master the subtle cause-and-effect relationship that exists between his/her movements and the surrounding fluid. This is what makes swimming one of the most technical of all sports. For the most part, science has played little if any role in helping swimmers and coaches improve swimming techniques or even to better understand the fluid dynamics of human swimming. Experiments of free swimming humans are extremely difficult to conduct and computational modeling approaches have, in the past, been unable to address this very complex problem. However, the development of a new class of numerical methods, coupled with unique animation and analysis tools is making it possible to analyze swimming strokes in all their complexity. The talk will focus on describing a relatively new numerical method that has been developed to solve flows with highly complex, moving/deforming boundaries. Numerical simulations are used to perform a detailed analysis of the dolphin kick. This stroke has emerged as an important component of competitive swimming in recent years and our analysis has allowed us to extract some useful insights into the fluid dynamics of this stroke. In addition, we also address the continuing debate about the role of lift versus drag in thrust production for human swimming.

11:48AM BW.00004 Baseball Aerodynamics: What do we know and how do we know it? , ALAN NATHAN, University of Illinois at Urbana-Champaign — Baseball aerodynamics is governed by three phenomenological quantities: the coefficients of drag, lift, and moment, the latter determining the spin decay time constant. In past years, these quantities were studied mainly in wind tunnel experiments, whereby the forces on the baseball are measured directly. More recently, new tools are being used that focus on measuring accurate baseball trajectories, from which the forces can be inferred. These tools include high-speed motion analysis, video tracking of pitched baseballs (the PITCHf/x system), and Doppler radar tracking. In this contribution, I will discuss what these new tools are teaching us about baseball aerodynamics.

12:14PM BW.00005 Fluid Mechanics of Cricket and Tennis Balls , RABINDRA D. MEHTA, Sports Aerodynamics Consultant, Mountain View, California — Aerodynamics plays a prominent role in defining the flight of a ball that is struck or thrown through the air in almost all ball sports. The main interest is in the fact that the ball can often deviate from its initial straight path, resulting in a curved, or sometimes an unpredictable, flight path. It is particularly fascinating that that not all the parameters that affect the flight of a ball are always under human influence. Lateral deflection in flight, commonly known as swing, swerve or curve, is well recognized in cricket and tennis. In tennis, the lateral deflection is produced by spinning the ball about an axis perpendicular to the line of flight, which gives rise to what is commonly known as the *Magnus effect*. It is now well recognized that the aerodynamics of sports balls are strongly dependent on the detailed development and behavior of the boundary layer on the ball's surface. A side force, which makes a ball curve through the air, can also be generated in the absence of the Magnus effect. In one of the cricket deliveries, the ball is released with the seam angled, which trips the laminar boundary layer into a turbulent state on that side. The turbulent boundary layer separates relatively late compared to the laminar layer on the other side, thereby creating a pressure difference and hence side force. The fluid mechanics of a cricket ball become very interesting at the higher Reynolds numbers and this will be discussed in detail. Of all the round sports balls, a tennis ball has the highest drag coefficient. This will be explained in terms of the contribution of the "fuzz" drag and how that changes with Reynolds number and ball surface wear. It is particularly fascinating that, purely through historical accidents, small disturbances on the ball surface, such as the stitching on cricket balls and the felt cover on tennis balls are all about the right size to affect boundary layer transition and development in the Reynolds numbers of interest. The fluid mechanics of cricket and tennis balls will be discussed in detail with the help of latest test data, analyses and video clips.

Sunday, November 22, 2009 2:00PM - 3:10PM –

Session C1 Prizes, Awards and Fellowships Presentations followed by Otto Laporte Lecture Auditorium

2:00PM C1.00001 Welcome, Presentation of Awards and DFD Fellowships –

2:25PM C1.00002 Otto Laporte Lecture: Fluid Dynamics Prize Talk: Simple Models for Turbulent Flows , STEPHEN B. POPE, Cornell University — We focus on the modeling of two turbulent flows: dispersion from a line source in grid turbulence; and, a lifted non-premixed turbulent jet flame. Stochastic Lagrangian models and PDF methods are described, and are shown to model these flows satisfactorily. For the line source, a Lagrangian approach is taken, with the Langevin equation modeling the velocity following a fluid particle, and with a simple relaxation model for the particle temperature. Comparison with experimental data shows that the resulting model describes accurately the dispersion from single and multiple line sources. These simple stochastic Lagrangian models are then applied to the much more challenging case of a lifted non-premixed jet flame. The stochastic Lagrangian models form the basis for a particle/mesh numerical method for solving a modeled transport equation for the Eulerian joint probability density function (PDF) of velocity and composition. The PDF calculations are in excellent agreement with the experimental data, and exhibit the observed extreme sensitivity of the flame to the temperature of the co-flow. The PDF model calculations presented clearly demonstrate that simple models can be very useful, even though aspects of their behavior may be inaccurate or incomplete. The shortcomings of the Langevin equation are examined, and more advanced models (designed to overcome some of these shortcomings) are described. These include models for fluid-particle acceleration, including the effects of intermittency; models accounting for mean shear, which are correct in the rapid- distortion limit; and models designed for use in conjunction with large-eddy simulations (LES).

Sunday, November 22, 2009 3:20PM - 3:55PM –
Session D1 Invited Session: Bio-mimetic Flow Control 205A-D

3:20PM D1.00001 Bio-mimetic Flow Control[1] , HAECHEON CHOI, Seoul National University — Bio-mimetic engineering or bio-mimetics is the application of biological methods and systems found in nature to the study and design of engineering systems and modern technology (from Wikipedia). The concept itself is old, but successful developments have been made recently, especially in the research field of flow control. The objective of flow control based on the bio-mimetic approach is to develop novel concepts for reducing drag, increasing lift and enhancing aerodynamic performance. For skin friction reduction, a few ideas have been suggested such as the riblet from shark, compliant surface from dolphin, microbubble injection and multiple front-body curvature from penguin, and V-shaped protrusion from sailfish. For form drag reduction, several new attempts have been also made recently. Examples include the V-shaped spanwise grooves from saguaro cactus, overall shape of box fish, longitudinal grooves on scallop shell, bill of swordfish, hooked comb on owl wing, trailing-edge protrusion on dragonfly wing, and fillet. For the enhancement of aerodynamic performance, focuses have been made on the birds, fish and insects: e.g., double layered feather of landing bird, leading-edge serration of humpback-whale flipper, pectoral fin of flying fish, long tail on swallowtail-butterfly wing, wing flapping motion of dragonfly, and alula in birds. Living animals adapt their bodies to better performance in multi purposes, but engineering requires single purpose in most cases. Therefore, bio-mimetic approaches often produce excellent results more than expected. However, they are sometimes based on people's wrong understanding of nature and produce unwanted results. Successes and failures from bio-mimetic approaches in flow control will be discussed in the presentation.

[1]Supported by the NRL Program of MEST.

Sunday, November 22, 2009 3:20PM - 3:55PM –
Session D2 Invited Session: Ocean Surface Waves and Turbulence: Air-Sea Fluxes and Climate Variability 208A-D

3:20PM D2.00001 Ocean Surface Waves and Turbulence: Air-Sea Fluxes and Climate Variability , W. KENDALL MELVILLE, Scripps Institution of Oceanography, University of California, San Diego — Apart from heating of the atmosphere, two of the most important consequences of current climate variability are changes in sea level, and acidification of the oceans. Over decadal time scales, changes in sea level are caused by changes in heat content and salinity of the ocean, and by changes in mass resulting from exchanges between the ocean, glaciers and other land-based reservoirs. The oceans have absorbed about one third of the anthropogenic CO_2 due to fossil fuel burning. This reduces the green house effect in the atmosphere, but the CO_2 reacts in the surface waters of the ocean to lower pH. Conservative projections of sea level rise over the next century are O(0.1 - 1) m, while ocean acidification is already having an impact on marine ecosystems. Both these processes depend on air-sea fluxes: heat flux for sea level rise, and gas flux for ocean acidification. These fluxes are among the most poorly constrained in current climate models, but both ultimately depend on fluid dynamics at the ocean surface and in the adjacent boundary layers. Traditional boundary layer models of the marine boundary layer and the marine atmospheric boundary layer were based on classical theories of boundary layers over rigid surfaces, but there is increasing evidence that these models must now include surface wave effects. In this talk the motivating climate data and modeling will be briefly reviewed, and then recent work on surface wave dynamics, air-sea fluxes and the adjacent boundary layers will be presented. The roles of surface wave breaking, Langmuir circulations, wave-turbulence interactions and gravity-capillary waves will be discussed.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session EA Turbulent Boundary Layers: Structure 101A

4:15PM EA.00001 A Method of Synthesizing 1-D POD Modes for Channel Flow[1] , JON BALTZER, RONALD ADRIAN, Arizona State University — POD analysis is performed on DNS channel data for several Reynolds numbers between $Re_\tau = 180$ and 950. The one-dimensional eigenmodes in the wall-normal direction are oscillatory and their phases can be extracted by various means. The phases contain the physics implied by the two-point spatial correlation, and the phases scaled using sequency correlate well with a single curve for all but the lowest mode numbers. An orthogonal basis to approximate the POD modes has been developed using this single phase function. Convergence using this basis is comparable to POD and superior to conventional orthogonal polynomials. The phase function is also compared to asymptotic results for large mode number (Moser, Phys. Fluids, **6**, 794-801, 1994).

[1]Supported by the Ira A. Fulton Endowment.

4:28PM EA.00002 Energetic modes in turbulent pipe flow from resolvent analysis , ATI SHARMA, Imperial College London, BEVERLEY MCKEON, California Institute of Technology — We describe a method to investigate the mode shapes in turbulent pipe flow at a given wavenumber pair that are most responsive to harmonic forcing. Specifically, these modes correspond to the largest singular values in a Schmidt decomposition of the linear Navier-Stokes operator using the turbulent mean profile as the base flow. The ideas follow logically from the work of Sharma & McKeon (2009), who considered a similar approach for laminar pipe flow. The dominant modes exhibit the inner and outer scaling behavior expected from experiment. A comparison of the mode shapes with the largest response with the results of a Karhunen-Loeve analysis on a pipeflow DNS by Duggleby *et al* (2007), i.e. an *a posteriori* evaluation of the most energetic mode shapes, shows reasonable agreement. An additional consequence of this analysis in terms of the scaling of the very large scale motions is discussed further in the presentation by McKeon & Sharma.

4:41PM EA.00003 Conditional structure of large- and small-scales in high Reynolds number turbulent boundary layers[1] , B. GANAPATHISUBRAMANI, Imperial College London, N. HUTCHINS, J.P. MONTY, I. MARUSIC, University of Melbourne — A spanwise array of surface hot-film shear-stress sensors and a traversing hot-wire located directly above one of these sensors are used to identify the conditional structure of a high Reynolds number turbulent boundary layer. The shear-stress data is low-pass filtered and is used to detect large-scale low and high skin-friction events. The velocity fluctuations from the hot-wire are decomposed in to large-scale and small-scale components and used to compute conditional large-scale mean velocity and small-scale turbulence intensities conditioned on the presence of large-scale low and high skin-friction events (detected by the skin-friction sensors). The conditional mean based on the large-scales show a forward-leaning low- and high-speed structure for low and high skin-friction events. The conditional small-scale turbulence intensity is weak near the wall and intense farther away from the wall for a low skin-friction event and vice-versa for a high skin-friction event. The changes in the velocity gradient of the conditional large-scale structure is used to explain the trends in the small-scale activity.

[1]BG acknowledges the Royal Society for the travel grant.

4:54PM EA.00004 DNS study of very-long coherent structures in turbulent pipe flow , RONALD ADRIAN, Arizona State University, XIAOHUA WU, Royal Military College of Canada, JON BALTZER, Arizona State University — Fully developed incompressible turbulent pipe flow at Reynolds number $Re_D = 24,580$ is simulated with second-order finite differences on a streamwise-periodic, 536 million point grid. The Karman number $R^+ = 648.8$, and the computational domain length is $30R$. The mean, second-order statistics and two-point correlations agree well with published experimental data. Pre-multiplied power spectra of the streamwise velocity peak at two wave numbers, one corresponding to very large scale motions (VLSM) 3–15R long, and the other corresponding to large scale motions (LSM) less than $3R$ long, consistent with earlier work (Kim and Adrian, Phys. Fluids **2**, 417–422, 1999, *et seq.*). The low speed patterns of VLS motion convect at the bulk velocity, and they are associated with strong, radially inward ejections from layers closer to the wall. They are made up of smaller structures having spacings of $\sim 0.2R = 125^+$ or less, coinciding with the typical spacing between hairpins in a near wall packet. The VLS motions also appear to leave a clear low-speed pattern in the near wall regions.

5:07PM EA.00005 The Structure of Large- and Very Large-Scale Motions in Turbulent Pipe Flow[1] , ALEXANDER SMITS, SEAN BAILEY, Princeton University — Multi-point velocity measurements in turbulent pipe flow have been performed at $Re_D = 1.5 \times 10^5$. Using cross-spectral and Proper Orthogonal Decomposition analysis, information is elucidated on the structure of the large- and very-large scale motions in the outer layer of wall-bounded flows. The results indicate that the large-scale motions are composed of attached eddies in the logarithmic layer but are mostly composed of detached eddies in the outer layer with a wide range of azimuthal scales. The very-large-scale motions have large radial and azimuthal scales, are concentrated around a single azimuthal mode, and make a smaller angle with the wall compared to the large-scale motions. The results support a hypothesis that only the detached large-scale motions in the outer layer align to form the very-large-scale motions.

[1]Supported by ONR through Grants No. N00014-07-1-0111 and N00014-09-1-0263 (program Manager Ron Joslin).

5:20PM EA.00006 Identification and analysis of boundary layer structures in Tomographic PIV data[1] , CECILIA ORTIZ-DUENAS, NEELAKANTAN SAIKRISHNAN, ELLEN LONGMIRE, Aerospace Engineering and Mechanics, University of Minnesota — Tomographic particle image velocimetry data were acquired in a turbulent boundary layer flow generated in a wind tunnel facility. The friction Reynolds number Re_τ was 1160. Measurement volumes of streamwise and spanwise dimension 0.7δ and wall normal thickness 0.07δ, which resolved the range z^+ = 70-150, were analyzed using custom software. Various criteria, including streamwise velocity, two- and three-dimensional swirl, individual vorticity components, and Reynolds shear stress, in combination with region growing and coincidence algorithms, were employed to identify and characterize coherent structures present in instantaneous fields. The results of the present study will be described and also compared with results from earlier experiments by Ganapathisubramani et al., 2003, 2006, that relied on planar PIV data only.

[1]Supported by NSF (CTS- 0324898).

5:33PM EA.00007 Hairpin packet structure of a turbulent boundary layer in inclined wall-normal/spanwise planes[1] , JAE HWA LEE, HYUNG JIN SUNG, KAIST — Turbulent coherent structures associated with hairpin packet motions have been scrutinized using the instantaneous flow fields obtained from the direct numerical simulation (DNS) of a turbulent boundary layer (TBL). The Reynolds number based on the momentum thickness was varied in the range Re_θ=890∼2560. This study focused on the hairpin packet motions in inclined wall-normal/spanwise planes. The hairpin vortex signature associated with the hairpin leg components in the vertical inclined plane consists of a counter-rotating vortex pair, upward and downward motions and a stagnation point induced by the Q2 and Q4 events. These hairpin signatures were observed in the instantaneous flow field, in the two-point correlations and in the conditionally averaged flow fields, respectively. We considered three inclined planes (45°, 90°, and 135°) to investigate the spatial characteristics of the hairpin packet motions in the log and wake regions. The statistical flow fields showed that significantly different flow patterns are induced by the intersections of the three inclined planes with the hairpin packet motions.

[1]This work was supported by KISTI under the Strategic Supercomputing Support Program.

5:46PM EA.00008 On the Existence of Hairpin Vortices , PETER BERNARD, University of Maryland — Numerical simulation of the transitioning and turbulent boundary layer using a vortex filament scheme (AIAA paper 2009-3547) provides a direct view of the organization of the vortex lines from which the structural aspects of the flow may be understood. It is found that the common assumption that the physical form of coherent vortical structures should be synonymous with regions of rotational motion is unwarranted. In fact, what are normally referred to as hairpin vortices are found not to be vortical objects in their own right, but rather the rotational motion corresponding to vortical features of a very different kind. The latter may be described as raised streamwise-oriented furrows in the vortex layer overlying low speed streaks. Upstream, the uplifted vorticity has a streamwise projection producing counter-rotating motion normally described as being the legs of hairpin vortices. Downstream, the upwelling vorticity detaches from the wall acquiring a mushroom shape, with the illusion of hairpin legs continuing within counter-rotating motion produced by streamwise oriented filaments. Arch vortices are produced by roll-up of the shear layers along the top of the furrows. The existence of hairpin vortices is equivalent to imagining that regions of rotational motion contain vorticity forming a complete structure - in contradiction to the objects observed in the filament simulation.

5:59PM EA.00009 The dynamical role of vortex tubes and sheets in wall-bounded flows , SERGIO PIROZZOLI, University of Rome "La Sapienza" — Vortex sheets and tubes are extracted from DNS of a canonical compressible boundary layer, and their dynamical contribution analyzed by means of a non-local analysis based on the solution of the Poisson equation for the vector potential. The results show non-negligible contribution of vortex sheets to the wall layer dynamics, especially in the inner layer. The statistical relationship between tubes and sheets is also analyzed by means of conditional average fields extracted from a DNS database. The results support strong association between the two types of coherent structures, and indicate that vortex tubes are mainly produced upon roll-up of vortex sheets (as in the hairpin vortex paradigm), or interact causing the ejection of near-wall vorticity, or generate sheets of streamwise vorticity through a rubbing effect caused by the no-slip condition.

6:12PM EA.00010 Hairpin vortex trajectories in a turbulent boundary layer , GERRIT ELSINGA, CHRISTIAN POELMA, JERRY WESTERWEEL, Delft University of Technology, ANDREAS SCHROEDER, REINHARD GEISLER, DLR Goettingen, FULVIO SCARANO, Delft University of Technology — Hairpin vortices within a turbulent boundary layer have been tracked in space and time resulting in their average trajectory and spreading rates with respect to the average. This information is used to predict the space-time correlation function of different flow variables, such as the individual velocity components and the invariants of the velocity gradient tensor, given their respective spatial auto-correlation functions. The predictions are in good agreement with the actual space-time correlations over convection distances up to one boundary layer thickness, which indicates that the decreasing peak correlation values over time reflect the spreading rate rather than an actual topological evolution of the individual flow structures. The results will also be discussed in relation to Taylor's hypothesis.

Sunday, November 22, 2009 4:15PM - 6:25PM –

Session EB Turbulence Modelling I 101B

4:15PM EB.00001 Modeling the pressure term in finite-volume LES with unresolved wall layers[1] , HENRY CHANG, ROBERT MOSER, University of Texas at Austin — In the incompressible Navier-Stokes equations, the fluctuating pressure term serves to keep the velocity field divergence-free. Here we consider the role of incompressibility and the pressure in LES with finite-volume filter and unresolved wall layers. In particular, we use a uniform staggered-grid finite-volume discretization, where $\Delta y^+ = 50$ in the wall-normal direction. The standard treatment of pressure in such a finite-volume discretization results in an LES field that satisfies a discrete divergence-free condition. However, solutions to the incompressible Navier-Stokes equations–when volume-averaged–are not discretely divergence-free. In fact, the discrete divergence increases sharply close to the wall. It is also found that the resulting discrete pressure does not represent well the pressure contribution to the evolution of the filtered Reynolds stress tensor. It appears, therefore, that the standard pressure solution is not appropriate for finite volume LES of wall-bounded turbulence. Using the techniques of optimal LES, alternative pressure models are being developed which: (1) do not strictly impose the discrete divergence-free condition and (2) correctly represent the contribution of pressure to the evolution of Reynolds stress.

[1]Financial support under AFOSR grant FA9550-07-1-0197 is gratefully acknowledged.

4:28PM EB.00002 Characterizing Finite Volume Operators for LES[1] , JEREMY HIRA, NICHOLAS MALAYA, VENKAT RAMAN, ROBERT MOSER, University of Texas at Austin — Optimal large eddy simulation (OLES) is an approach to LES sub-grid modeling that requires multi-point correlation data as input. Until now, this has been obtained by analyzing DNS statistics. In the finite-volume OLES formulation studied here, under the assumption of small-scale homogeneity and isotropy, these correlations can be theoretically determined from Kolmogorov inertial-range theory and the quasi-normal approximation. Resulting models are expressed as generalized quadratic and linear finite volume operators that represent the convective momentum flux, and they have been found to produce accurate LES results. The operators have been analyzed to determine their characteristics as numerical approximation operators and as models of subgrid effects. In addition, the dependence of the model operators on the anisotropy of the grid and on the size of the stencils is analyzed to develop idealized general operators that can be used on general grids. These constitute finite volume turbulence operators applicable in a wide range of LES problems.

[1]Support under NASA gran NNX08AD03A is gratefully acknowledged.

4:41PM EB.00003 A grid-independent length scale for large-eddy simulations , U. PIOMELLI, Queen's University, Canada, B.J. GEURTS, University of Twente, Netherlands — In most large-eddy simulations a length-scale related to the grid size is used in the subgrid-scale models. Rapid variations of the mesh may cause errors and unphysical results. We propose a new length scale for small-scale turbulence models that is decoupled from the grid, and is determined dynamically from the velocity field itself. It is based on an approximation to a local integral scale used in turbulence models. The resulting eddy-viscosity model has many features of dynamic models (it vanishes near a wall or in laminar flows, and senses the local small scales of the flow) but does not require the use of spatial filtering operations, which are costly and may be difficult to perform on unstructured grids. The model coefficient is determined by a Successive Inverse Polynomial Interpolation procedure (Geurts & Meyers, 2006), in which the coefficient is optimized computationally to minimize a specified cost function. Since this procedure can be performed on coarse grids, it adds little to the computational cost of the method. A set of 4-6 coarse simulations with the new model is required to approximate the optimum with fair accuracy, and the total cost of a simulation is comparable to that of a single simulation with a dynamic model. The new length scale also has the desirable feature that refining the mesh does not result in a DNS, but in a grid-converged LES. Applications to plane channel and mixing layers will be presented.

4:54PM EB.00004 A modulated gradient model for large-eddy simulation: application to a neutral atmospheric boundary layer[1] , HAO LU, FERNANDO PORTE-AGEL, Dept. Civil Engr., UMN — It is known that in LES of high-Re atmospheric boundary layer (ABL), standard eddy-viscosity models poorly predict mean shear in the near-wall region and yield erroneous velocity profiles. A modulated gradient model is proposed. It is based on the Taylor expansion of the SGS stress and uses local equilibrium hypothesis to evaluate the SGS kinetic energy. To ensure numerical stability, a clipping procedure is used to avoid local backscatter. Two approaches are considered to specify the model coefficient: a constant value of 1, and a simple correction to account for the effects of the clipping procedure on the SGS energy production rate. The model is assessed through a systematic comparison with well-established empirical formulations and theoretical predictions of a variety of flow statistics in a neutral ABL. The statistics of the simulated velocity field obtained with the model show good agreement with reference results and a significant improvement compared to simulations with standard models. For instance, it is capable to reproduce the expected log-law profile and power-law energy spectra. It also yields streaky structures and near-Gaussian PDFs of velocity in the near-wall region.

[1]This project was supported by the NSF, NASA, Xcel Energy, and IREE. Computing resources were provided by MSI.

5:07PM EB.00005 Progress Towards Specifying Initial Conditions for Variable Density Turbulence Models , BERTRAND ROLLIN, MALCOLM J. ANDREWS, Los Alamos National Laboratory — It is now well accepted in the turbulence community that variable density turbulence can be affected at late time by the initial perturbations. This important property has opened an opportunity for prediction and "design" of late-time turbulence of particular interest for many engineering purposes. Specifically, our study aims at defining the rules for properly accounting for the initial conditions in variable density turbulence models. We report our latest advancement in this direction. A nonlinear ODE model is used to compute the evolution of bubbles/spikes in Rayleigh-Taylor/Richtmyer-Meshkov mixing after carefully formulated initial perturbations. We investigate the relations between the composition of the initial conditions, key characteristics, and quantities of the bubble/spikes evolution. Properties and possible scalings that dictate the late-time behavior of the flow will be discussed.

5:20PM EB.00006 Recovery of subgrid-scale kinetic energy in large-eddy simulations of incompressible wall-bounded flows , YIFENG TANG, RAYHANEH AKHAVAN, Dept. of Mechanical Engineering, University of Michigan, Ann Arbor, MI 48109-2125, USA — A method is presented for recovering the subgrid-scale kinetic energy in large-eddy simulations (LES) of wall-bounded flows. The formulation is based on extending the one-dimensional energy spectra obtained in LES using the filtered one-dimensional energy spectra derived from the theoretical formulations of Pao (1965) or Meyers and Meneveau (2008) for the three-dimensional energy spectrum in isotropic turbulence. To allow for application of these formulations to wall-bounded flows, the LES spectra are re-normalized into an isotropic space. Once the SGS kinetic energy is recovered, the individual components of turbulence intensities are computed using the formulation of Winckelmans *et al.* (2002). The entire procedure is applied as a post-processing step and can be combined with any SGS model. In tests performed using filtered DNS databases of turbulent channel flow at a $Re_\tau \approx 570$, the method recovered the SGS kinetic energy with errors of less than 10% and the total kinetic energy with errors of less than 1%. In application to LES data obtained using the Dynamic Smagorinsky Model, the individual components of turbulence intensities were recovered with an accuracy comparable to that with which the filtered statistics were predicted in LES.

5:33PM EB.00007 SGS model based on spatial correlation between turbulence structure and energy transfer , HIROMICHI KOBAYASHI, Keio University — In LES, it is well known that the spatial correlation between the energy transfer term obtained from filtered DNS and that from the Smagorinsky model is rather poor, especially in channel flows. Recently, several references showed that the local equilibrium proposed by Smagorinsky between the subgrid-scale (SGS) energy production and the SGS energy dissipation in the SGS kinetic energy equation seems to be not valid. The energy transfer from grid-scale (GS) to SGS is called the forward scatter. The energy transfer term is often called the SGS energy production term. The spatial correlation between the energy transfer term with the forward scatter and turbulence structure is examined. It is found that the negative second invariant of a velocity gradient tensor surrounding a vortex gives relatively high correlation in not only homogeneous turbulence but also a channel flow. A SGS model based on the correlation is proposed.

5:46PM EB.00008 Possible modifications to implicit large-eddy simulation , J.M. MCDONOUGH, University of Kentucky — Implicit large-eddy simulation (ILES) provides an advantage over more usual LES approaches in that its construction does not involve filtering of the governing equations and, as a consequence, removal of the need to develop sub-grid scale (SGS) models to represent artificial stresses arising from this filtering. At the same time, it is clear that ILES is simply an under-resolved direct numerical simulation with advanced treatments of advection terms to better control numerical stability via dissipation that otherwise would have been provided by a SGS model. As such it cannot be expected to accurately predict interactions of fluid turbulence with other physical phenomena ($e.g.$, heat and mass transfer, chemical kinetics) on subgrid scales—as is also true of usual forms of LES. In this talk we describe a straightforward technique, based on formal multi-scale methods, whereby SGS interactions can be introduced to enhance resolved-scale results computed as in ILES, and we discuss derivation of a class of efficient models based on the "poor man's Navier–Stokes equation" (McDonough, $Phys.Rev.\ E$ **79**, 2009; McDonough and Huang, $Int.J.Numer.\ Meth.\ Fluids$ **44**, 2004). Properties of these models will be presented for a moderate-Re 3-D lid-driven cavity problem.

5:59PM EB.00009 Evaluation of Leray, LANS and Verstappen regularizations in LES, without and with added SGS modeling , G. WINCKELMANS, N. BOURGEOIS, Y. COLLET, M. DUPONCHEEL, UCL — Regularization approaches (Leray, LANS and Verstappen) for the "restriction in the production of small-scales" in turbulence simulations have regained some interest in the LES community, because of their potentially appealing properties due to filtering. Their potential is here investigated using the best possible numerics (dealiased pseudo-spectral code) and on simple problems: transition of the Taylor-Green vortex (TGV) and its ensuing turbulence, developed homogeneous isotropic turbulence (HIT). The filtered velocity field is obtained using discrete filters, also of various orders (2 and 6). Diagnostics include energy, enstrophy, and spectra. The performance of the regularizations on the TGV is first evaluated in inviscid mode (96^3 Euler), then in viscous mode at $Re = 1600$ (256^3 DNS and 48^3 LES). Although they delay the production of small scales, none of the regularizations can perform LES when the flow has become turbulent: the small scales are still too energized, and thus added subgrid-scale (SGS) modeling is required. The combination of regularization and SGS modeling (here using the RVM multiscale model) is then also evaluated. Finally, 128^3 LES of fully developed HIT at very high Re is also investigated, providing the asymptotic behavior. In particular, it is found that the regularization helps increase a bit the true inertial subrange obtained with the RVM model.

6:12PM EB.00010 On Closure Model for Accurate Reduced-Order Modeling in 3-D Flows , IMRAN AKHTAR, JEFF BORGGAARD, TRAIAN ILIESCU, ZHU WANG, Virginia Tech — Reduced-order models based on the proper orthogonal decomposition (POD) can be used to represent and understand complex dynamical systems such as the Navier-Stokes equations. These models give insight to the flow physics, reproduce the data, and may be used for control purposes. However, most successful applications of this approach involve low Reynolds number, 2-D flows. For most 3-D flows, a large number of POD modes are required to accurately represent the flow field. The large dimension of the resulting model contradicts the essence of model reduction. Furthermore, the resulting reduced-order model is usually numerically unstable. In this study, we suggest an LES-type closure model within the POD-based reduced-order modeling framework. We simulate the flow past a 3-D cylinder at Re=1000 and collect a large set of snapshots to capture turbulent structures in the wake. We develop a reduced-order model using a small number of POD modes and introduce an additional term within the model to capture the effects of high frequencies (e.g. discarded modes) in the system. We compare the results of the model with the DNS data to establish accuracy of the modified reduced-order model.

Sunday, November 22, 2009 4:15PM - 5:59PM –
Session EC Turbulence Simulations III 101C

4:15PM EC.00001 Large Eddy Simulation of Cavitation Inception in a High Speed Flow Over an Open Cavity[1] , EHSAN SHAMS, SOURABH APTE, Oregon State University — Large-eddy simulation of flow over an open cavity corresponding to the experimental setup of Liu and Katz [Liu & Katz, PoF 2008] is performed using a co-located grid finite-volume solver. The flow Reynolds number based on the cavity length and the free stream velocity is $170,000$. The flow statistics, including mean and rms velocity fields and pressure coefficients, are compared with the experimental data to show good agreement. Cavitation inception was investigated using two approaches: (i) a discrete bubble model for gaseous cavitation based on solving the Rayleigh-Plesset equations using an adaptive time-stepping procedure, and (ii) a scalar transport based model for vaporous cavitation. Sensitivity to the model parameters was investigated by varying the model parameters and by changing the cavitation index. Both models predict that the cavitation inception occurs near the trailing edge similar to that observed in the experiments. A periodic growth and decay of bubble size and liquid vapor fraction is observed above the trailing edge owing to local variations in pressure minima. The probablity distribution functions and average number of bubbles undergoing cavitation predict an inception index of 0.9 that agrees well with the experimental data.

[1]This work was supported by ONR Grant N000140610697.

4:28PM EC.00002 Large Eddy Simulation of Ducted Propulsors in Crashback[1] , HYUNCHUL JANG, KRISHNAN MAHESH, Aerospace Engineering, University of Minnesota — Flow around a ducted marine propulsor is computed using the large eddy simulation methodology under crashback conditions. Crashback is an operating condition where a propulsor rotates in the reverse direction while the vessel moves in the forward direction. It is characterized by massive flow separation and highly unsteady propeller loads, which affect both blade life and maneuverability. The simulations are performed on unstructured grids using the discrete kinetic energy conserving algorithm developed by Mahesh at al. (2004, J. Comput. Phys 197). Numerical challenges posed by sharp blade edges and small blade tip clearances are discussed. The flow is computed at the advance ratio J=-0.7 and Reynolds number Re=480,000 based on the propeller diameter. Average and RMS values of the unsteady loads such as thrust, torque, and side force on the blades and duct are compared to experiment, and the effect of the duct on crashback is discussed.

[1]This work was supported by the United States Office of Naval Research under ONR Grant N00014-05-1-0003.

4:41PM EC.00003 Explicit Wavelet Filtering in Stochastic Coherent Adaptive Large Eddy Simulation[1] , GIULIANO DE STEFANO, Seconda Universitá Napoli, Italy, OLEG V. VASILYEV, University of Colorado at Boulder — The Stochastic Coherent Adaptive Large-Eddy Simulation (SCALES) method is a novel approach to the numerical simulation of turbulence, where the coherent energetic eddies are solved while modeling the effect of the less energetic background flow. In the explicit-filtering approach, additional explicit wavelet thresholding filter is applied, along with the implicit filter induced by the use of the adaptive wavelet collocation method to solve the governing equations. Two different thresholding levels are clearly identified: the physical threshold that controls the formal separation between resolved coherent eddies and residual coherent/incoherent flow, and the numerical threshold that controls the numerical accuracy of the method. A number of numerical experiments is conducted to study the effect of the numerical thresholding level on the accuracy and computational efficiency of the SCALES method and the trade-off between modeling and numerical issues. The explicit wavelet filtering allows us to analyze the quality of SCALES solutions with respect to ideal grid-independent solutions, enhancing our knowledge about the strong interaction that exists between wavelet grid-compression and modeled turbulent dissipation.

[1]This work was supported by NSF under grant No. CBET-0756046.

4:54PM EC.00004 ABSTRACT WITHDRAWN –

5:07PM EC.00005 ABSTRACT WITHDRAWN –

5:20PM EC.00006 Numerical studies on the rise of microscopic oil droplets in high intensity isotropic turbulence , MURRAY SNYDER, United States Naval Academy — The rise of small oil droplets in water under three different isotropic turbulence conditions is analyzed. The simulations focus on explaining the puzzling behavior observed by both Friedman and Katz [Phys. Fluids **14**, 3059 (2002)] and Gopalan *et al.* [Phys. Fluids **20**, 095102 (2008)], specifically, the size dependent enhanced or suppressed rise of small oil droplets in turbulence. Both showed that droplets with diameters smaller than approximately 900 microns exhibited enhanced rise when compared with quiescent rise behavior. Conversely, they observed that larger droplets exhibited retarded rise in turbulence versus quiescent conditions. Snyder *et al.* [Phys. Fluids **20**, 073301 (2008)] showed that the experimental results of Friedman and Katz could be captured using a 128^3 direct numerical simulation with a dynamical droplet equation of motion. Snyder *et al.*, however, used non-physical approximations that drag and virtual mass coefficients depend upon mean turbulence intensity or droplet size. Enhanced computations have been done with both 128^3 and 1024^3 direct numerical simulations using the commonly observed correlation that drag and virtual mass coefficients vary with droplet Reynolds number. These computations show that the observed experimental behavior can be approximately captured using Reynolds number dependent drag and virtual mass coefficients for Kolmogorov microscales of 60-180 microns.

5:33PM EC.00007 Direct simulation of fully-developed turbulent flow bounded by perfectly-permeable wall , SATOSHI YOKOJIMA, Shizuoka University, Japan — The effect of wall imperviousness (wall-blocking effect) on the turbulent channel flows has been investigated. To this end, we numerically realize a new system, fully-developed turbulent flow bounded by a perfectly-permeable wall which is obtained by removing only the impermeable properties from a solid wall. It is shown that the perfectly-permeable wall has a drag two-order-of-magnitude higher than does the impermeable solid wall, indicating that permeable boundaries can be an efficient mixing device.

5:46PM EC.00008 Vorticity Based Turbulence Model Applied to an Impulsively Moved Flat Plate , NICHOLAS KACHMAN, U.S. Navy NAVAIR — A novel technique to model turbulence by vorticity in solid body rotation is presented. The model is based on simultaneously solving the vorticity equation and the Navier-Stokes equation for a 2-D unsteady boundary layer. Only that vorticity that is in solid body rotation is used to develop perturbation velocities that are then applied to the unsteady boundary layer equations. New vorticity is introduced each time step, when the vorticity equation produces a value different from that calculated by the boundary layer equations. Comparing the numerical results to experimental flow visualization demonstrates similar characteristic traits to a turbulent boundary layer, such as no turbulence until Re $\sim$ 4.5x10^4, intermittency, velocity "tubes" that ejected fluid into and out of the boundary layer, and turbulent decay after leaving the plate. Issues remain with the method. First, the velocity perturbations and boundary layer growth are less than expected. It is believed that this is due to the 2-D nature of the solution and that the move to 3-D and the incorporation of vortex stretching will provide values closer to experimental results. Second, the velocity perturbations cause the mesh Reynolds number to be exceeded, which needs to be addressed in future work.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session ED Flow Control III 101D

4:15PM ED.00001 Development of Reduced-order Models for Feedback Control of Axisymmetric Jets[1] , ANIRUDDHA SINHA, ANDREA SERRANI, MO SAMIMY, Ohio State University — We present the preliminary steps toward development of reduced-order models (ROM) for feedback control of a high-speed and high Reynolds number axisymmetric jet. The control objective is two-fold: attenuation of far-field acoustic radiation, or, enhancement of bulk mixing, using a set of localized arc filament plasma actuators that perturb the initial shear layer of the jet through intense localized Joule heating. The proposed feedback sensing mechanism involves pressure information from the irrotational near-field of the jet. The proposed route for creating the ROM involves collecting PIV data of the jet simultaneously with the pressure measurements, performing Proper Orthogonal Decomposition and Stochastic Estimation to obtain a time- and space-resolved database, and using Galerkin Projection to derive the dynamical model. Here we evaluate the above strategy using a DNS database (Freund, J. B., J. Fluid Mech., 438, 2001, 277–305). The ROMs obtained using various modeling options are simulated and their comparative fidelity are adjudged based on the original simulation results.

[1]Supported by AFOSR.

4:28PM ED.00002 Hybrid Ensemble and Variational Estimation for Chaotic Systems , JOSEPH CESSNA, THOMAS BEWLEY, University of California San Diego — An estimate of a system is developed from knowledge of the model of the system along with a time history of noisy measurements. Unlike a filtered result, an estimate conditioned on both past and future measurements is said to be a smoothed estimate. For linear systems, computing the prior smoothed estimate trajectory appropriately has no bearing on the most recent filtered estimate. However, in chaotic systems, the estimate probability distribution is non-gaussian and must be computed through some form of linearization. As a result, one would expect that knowing a prior trajectory more accurately (through smoothing) could be beneficial in improving the most recent estimate via a reinterpretation of past measurements about this new trajectory. Unfortunately, for high-dimensional chaotic systems, typical smoothing approaches such as the Ensemble Kalman Smoother update the prior smooth trajectories upon receipt of a new measurement without altering the most recent filtered estimate. Using a new hybrid approach combining the EnKS with an adjoint variational method, we examine how the smoothed trajectories can be used consistently to improve the most recent filtered estimate. By appropriately defining a cost function, we can use the EnKS estimate as an initial condition for a well-posed variational iteration. It is shown that this technique reduces to the well known optimal Kalman Smoother result in the case of a linear system.

4:41PM ED.00003 State estimation of turbulent channel flow , CHRISTOPHER COLBURN, THOMAS BEWLEY, UC San Diego — The challenges involved in state estimation of wall-bounded turbulent flows are considered through computational experiments using a variety of measurement strategies (including wall information only). These experiments are part of a larger ongoing effort to develop high fidelity estimates of turbulent flows using the Ensemble Kalman Filter, a method used widely within the weather forecasting community, and other related estimation strategies. This study, which attempts to quantify rigorously the "propagation of information" in turbulent flows, might also help to shed some new light on various "top-down" versus "bottom-up" hypotheses currently being debated in the literature on near-wall and highly-sheared turbulent flows.

4:54PM ED.00004 EnVO: a hybrid ensemble/variational adaptive observation algorithm , DAVID ZHANG, THOMAS BEWLEY, UCSD — Advances in autonomous technology enable the development of Adaptive Observation (AO) strategies, which identify future sensor locations to reduce forecast error. The approaches to AO strategies are mainly divided into two techniques, the "uncertainty" based techniques such as the Ensemble Transform Kalman Filter by Bishop et al. 2001 and the "sensitivity" based techniques such as the Singular Vector by Palmer et al. 1998. Both approaches have their advantages and disadvantages; hence the combination of both is perhaps ideal. We propose a hybrid ensemble/adjoint AO technique, dubbed Ensemble Variational Observation (EnVO). The EnVO technique uses *both* sensitivity and uncertainty information, and can be applied in such a way as to ensure its optimized sensor trajectories are both feasible and efficient over the time window between the present and forecast time. Integrated statistic results indicates EnVO is able to consistently identifies target sites that produce lower forecast errors.

5:07PM ED.00005 Grid-based Bayesian Estimation Exploiting Sparsity for systems with non-gaussian uncertainty , THOMAS BEWLEY, UC San Diego, ATI SHARMA, Dept of Aeronautics, Imperial College — We present a new algorithm for Bayesian estimation of nonlinear ODE systems $d\mathbf{x}/dt = \mathbf{f}(\mathbf{x})$ with finite, nongaussian uncertainty. The algorithm presented represents the evolution of the probability distribution in phase space, $P(\mathbf{x}, t)$, discretized on an Eulerian (that is, fixed, Cartesian) grid, and consists of two main steps: (1) Between measurement times, $P(\mathbf{x}, t)$ is marched via careful numerical discretization of the PDE governing its evolution using a Godunov method with second-order CTU correction and an MC flux limiter. (2) At measurement times, $P(\mathbf{x}, t)$ is updated via Bayes' theorem. The key to the efficiency of the new method is a novel technique for leveraging *sparsity* of the probability distribution (that is, leveraging the fact that it is essentially zero almost everywhere in phase space). The absence of a fundamental dependence on a central estimate and the second-order moments of its uncertainty renders the new approach better suited than Kalman-based approaches to nongaussian uncertainty distributions, while the Eulerian discretization of $P(\mathbf{x}, t)$ in the new approach avoids the sticky wicket associated with Lagrangian ("particle"-based) discretizations.

5:20PM ED.00006 Global Model Reduction for Fluid-Structure Interaction in Flapping Flexible Wings[1] , MINGJUN WEI, TAO YANG, New Mexico State University — Reduced-order models (ROMs) for fully-coupled fluid-structure interaction problems are desired in many applications (e.g. design of flapping-wing Micro Air Vehicles). Traditional approach is to build ROMs individually for fluid and solid and couple them through the interface. In this work, we suggest an approach to apply model reduction globally on a uniform description of fluid and solid in Eulerian framework. The idea has been made possible by a set of combined fluid-structue equations, where solid properties are presented as extra terms to Navier-Stokes equations. Then, typical Proper Orthogonal Decomposition (POD)/Galerkin projection can be used for model reduction as in most fluid-only problems, with special care of the extra "solid" terms. In the example, we show that one can capture most energy by only a few POD modes. More importantly, the leading POD modes show the signatures of both fluid flow and solid structure.

[1]The simulation is supported by Army High Performance Computing Research Center.

5:33PM ED.00007 Application of Proper Orthogonal Decomposition to Disk Wakes , ZACHARY BERGER, AARON J. ORBAKER, MARK N. GLAUSER, HIROSHI HIGUCHI, MAKAN FARDAD, RORY BIGGER, Syracuse University — This work investigates the effects of flow control on the near wake region of a disk in a water flow utilizing the POD reconstructed time dependent velocity fields. Velocity measurements were collected using time resolved particle image velocimetry (TRPIV) at a Reynolds number of 20,000 based on the disk diameter both with and without control. Since this is a time resolved velocity data base we are able to reconstruct the time dependent velocity field in the wake for baseline and controlled cases using various POD truncations and observe velocity reconstructions. The current interest is in the convergence of the spatial eigenvalues of the baseline and controlled cases for the future development of closed-loop control systems from the perspective of the time dependent velocity field instead of just using the normal energy convergence criteria.

5:46PM ED.00008 Frequency-Domain Proper Orthogonal Decomposition of Synthetic-Jet Actuated Flow for Airfoil Control[1] , GUY BEN-DOV, ARNE J. PEARLSTEIN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, DANIEL P. BRZOZOWSKI, ARI GLEZER, Woodruff School of Mechanical Engineering, Georgia Institute of Technology — Using a proper orthogonal decomposition in the *frequency domain*, we construct a forced ODE system for use as a reduced-order model of flow over an actuated airfoil. The time-dependent part of the flow in the wake, attributable to actuation, is linearized about the mean wake flow (generated from time-averaged flow fields obtained from PIV data). The resulting PDE system consists of the continuity and linearized vorticity transport equations. Using instantaneous inlet conditions from several realizations with different actuation, we apply a spectral method to compute the impulse response of the PDEs in the frequency domain over a wide range of frequency. The decomposition allows substitution of the resulting modes into the PDEs, and Galerkin projection to ODEs. Only the forcing terms are frequency-dependent, and they can be parametrized on an actuator-specific basis. A neural network technique is suggested to relate the dynamics (given by the ODEs) to the aerodynamic forces on the airfoil, which will allow the model to be used as the basis of a control system.

[1]Supported by AFOSR Grant FA9550-05-1-0411.

5:59PM ED.00009 Active Control of Aerodynamic Forces on a Rapidly Maneuvering Airfoil[1] , DANIEL BRZOZOWSKI, JOHN CULP, ARI GLEZER, Georgia Insitute of Technology — The unsteady aerodynamic forces and moments on a rapidly maneuvering free-moving airfoil are investigated in wind tunnel experiments. The airfoil is mounted on a 2-DOF traverse and its trim and dynamic characteristics are controlled using position and attitude feedback loops that are actuated by servo motors. The motion of the airfoil is effected by bi-directional changes in the pitching moment using controllable trapped vorticity concentrations on both the suction and pressure surfaces near the trailing edge that are induced and regulated by hybrid synthetic jet actuators. 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 unsteady flow characteristics induced by the fluidic actuation during a prescribed maneuver are compared with the effects of a simple rigid-body motion of the airfoil when an external torque is used to achieve a similar maneuver. It is shown that the time-dependent aerodynamic forces and induced flow fields in the near wake of the moving airfoil are significantly different, emphasizing the role of the coupling between the flow control actuation and the model's unsteady aerodynamics.

[1]Supported by AFOSR-MURI.

6:12PM ED.00010 Effects of Oscillatory Actuation Frequency on Wall-Mounted Hump Flow , JENNIFER FRANCK, Brown University, TIM COLONIUS, California Institute of Technology — A large eddy simulation (LES) is used to explore the effects of high frequency actuation on the natural separation bubble formed by a wall-mounted hump. Low frequency forcing at $F^+ \sim O(1)$ has been shown to increase entrainment through regular shedding of large-scale structures. Using a LES technique previously validated on the baseline and controlled flow over the wall-mounted hump geometry, the effectiveness of high frequency actuation is explored and compared with previous investigations. It is found that the high frequency actuation does not produce distinct vortical structures in the separated shear layer, and does not delay the onset of separation. The resulting mean flow is relatively unaltered by the application of high frequency actuation, although the local flow surrounding the actuation location is slightly modified.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session EE Biofluids III: General III - Flows and Diseases 101E

4:15PM EE.00001 Probing protein mechanical stability with controlled shear flows , JONATHAN DUSTING, Experimental and Computational Laboratory for the Analysis of Turbulence (ECLAT), King's College London, Strand, London WC2R 2LS, UK, LORNA ASHTON, Manchester Interdisciplinary Biocentre, University of Manchester, Manchester, M1 7DN, UK, JUSTIN LEONTINI, FLAIR, Dept of Mechanical and Aerospace Engineering, Monash University, Australia, EWAN BLANCH, Manchester, UK, STAVROULA BALABANI, London, UK — Understanding and controlling protein aggregation or misfolding is of both fundamental and medical interest. The structural changes experienced by proteins in response to forces such as those generated within flows have not been well characterised, despite the importance of mechanics in many biological processes. By monitoring the structural conformation of proteins in different concentric cylinder flows using Raman Spectroscopy we have quantified the relative stability of β-sheet dominated proteins compared with those containing a greater proportion of α-helix. To ensure that the fluid stresses are quantified accurately, a combined DNS and PIV approach has been undertaken for flow cell characterisation across the full range of operating Re. This is important for practical concentric cylinder geometries where the shear components are non-zero and spatially dependent, with the peak stresses located near the endwalls. Furthermore, recirculation regions appear well below the crtical Reynolds number for Taylor vortex formation.

4:28PM EE.00002 A quasi-one-dimensional model for collapsible channel oscillations , DRAGA PIHLER-PUZOVIC, TIMOTHY PEDLEY — A fluid driven rapidly through a flexible tube exhibits self-excited oscillations. To model this phenomenon, we consider 2D high Re laminar flow of a Newtonian incompressible fluid through a collapsible channel. The channel has a section of an otherwise rigid wall replaced by a membrane with inertia, under longitudinal tension, with no bending stiffness and subject to the external pressure. Based on the analysis by Pedley and Stephanoff (*JFM*,85), membrane motion is coupled to the time-dependent behaviour of the core flow through a modified KdV equation. We focus on the importance of membrane inertia for the system. The stability of the problem is studied numerically. In the parameter regimes of interest the computations reveal transitional behaviour: initially small perturbation of the system decays in an oscillatory manner but beyond a certain time higher frequency oscillations start dominating and the system diverges. At the same time a switching between mode one in which the flexible wall has a single extremum, to higher modes with multiple extrema is observed. These results are discussed with respect to previous computations for 2D collapsible channels.

4:41PM EE.00003 Phasic Relationships among Hemodynamic Properties of Pulsatile Flow in Microcirculations[1] , JUNG YEOP LEE, SANG JOON LEE, Dept. of Mech. Eng., POSTECH, Republic of Korea, BIOFLUID AND BIOMIMIC RESEARCH CENTER TEAM — Pulsatile blood flows in *omphalo-mesenteric* arteries of HH-stage 18 chicken embryos are measured using a time-resolved particle image velocimetry (PIV) technique to obtain hemodynamic information in microcirculations and compare hemodynamic properties of pulsatile blood flows. Due to the intrinsic features of pulsatile flow and complicated vessel network of microcirculation, an *out-of-phase* motion of blood occurs in nearby vessel segments of bifurcations. This is mainly attributed to the morphological characteristics and peripheral resistance of vasculature. The *out-of-phase* motion is quantitatively identified using the one-dimensional temporal cross-correlation function. This cross-correlation function is extended to establish the phasic relationships among hemodynamic properties such as velocity, shear rate, and acceleration. Velocity and shear rate are almost *in phase*, as predicted theoretically. On the other hand, velocity (or shear rate) shows an almost 180° *out-of-phase* against acceleration, which is quite larger than the theoretically predicted value.

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

4:54PM EE.00004 Numerical Study on Flows of Red Blood Cells with Liposome-Encapsulated Hemoglobin at Microvascular Bifurcation , TORU HYAKUTAKE, Yokohama National University, SHIGEKI TANI, YUKI AKAGI, Okayama University, TAKESHI MATSUMOTO, Osaka University, SHINICHIRO YANASE, Okayama University — Flow analysis at microvascular bifurcation after partial replacement of red blood cell (RBC) with liposome-encapsulated hemoglobin (LEH) was performed using the lattice Boltzmann method. A two-dimensional bifurcation model with a parent vessel and daughter branch was considered, and the distributions of the RBC, LEH, and oxygen fluxes were calculated. The immersed boundary method was employed to incorporate the fluid–membrane interaction between the flow field and deformable RBC When only RBCs flow into the daughter branches with unevenly distributed flows, plasma separation occurred and the RBC flow to the lower-flow branch was disproportionately decreased. On the other hand, when the half of RBC are replaced by LEH, the biasing of RBC flow was enhanced whereas LEH flowed favorably into the lower-flow branch, because many LEH within the parent vessel are suspended in the plasma layer, where no RBCs exist. Consequently, the branched oxygen fluxes became nearly proportional to flows. These results indicate that LEH facilitates oxygen supply to branches that are inaccessible to RBCs.

5:07PM EE.00005 A Numerical Computation Model for Low-Density Lipoprotein (LDL) Aggregation and Deposition in the Human Artery , YONGLI ZHAO, SCSU, SHAOBIAO CAI, PSU, ALBERT RATNER, University of Iowa — Cholesterol caused cardiovascular events are commonly seen in human lives. These events are primarily believed to be caused by the built up of particles like low-density lipoprotein (LDL). When a large number of LDL circulates in the blood, it can gradually build up in the inner walls of the arteries. A thick, hard deposit plaque can be formed together with other substances. This type of plaque may clog those arteries and cause vascular problems. Clinical evidences suggest that LDL is related to cardiovascular events and the progression of coronary heart disease is due to its aggregation and deposition. This study presents an investigation of LDL aggregation and deposition based on particulate flow. A soft-sphere based particulate computational flow model is developed to represent LDL suspending in plasma. The transport, collision and adhesion phenomena of LDL particles are simulated to examine the physics involved in aggregation and deposition. A multiple-time step discrete-element approach is presented for efficiently simulating large number of LDL particles and their interactions. The roles the quality and quantity the LDL playing in the process of aggregation and deposition are determined. The study provides a new perspective for improving the understanding of the fundamentals as related to these particle-caused cardiovascular events.

5:20PM EE.00006 A Comprehensive Fluid Dynamic-Diffusion Model of Blood Microcirculation with Focus on Sickle Cell Disease , FRANCOIS LE FLOCH, WESLEY L. HARRIS, MIT — A novel methodology has been developed to address sickle cell disease, based on highly descriptive mathematical models for blood flow in the capillaries. Our investigations focus on the coupling between oxygen delivery and red blood cell dynamics, which is crucial to understanding sickle cell crises and is unique to this blood disease. The main part of our work is an extensive study of blood dynamics through simulations of red cells deforming within the capillary vessels, and relies on the use of a large mathematical system of equations describing oxygen transfer, blood plasma dynamics and red cell membrane mechanics. This model is expected to lead to the development of new research strategies for sickle cell disease. Our simulation model could be used not only to assess current researched remedies, but also to spur innovative research initiatives, based on our study of the physical properties coupled in sickle cell disease.

5:33PM EE.00007 Investigating the fluid mechanics behind red blood cell-induced lateral platelet motion , LINDSAY CROWL ERICKSON, AARON FOGELSON, University of Utah — Platelets play an essential role in blood clotting; they adhere to damaged tissue and release chemicals that activate other platelets. Yet in order to adhere, platelets must first come into contact with the injured vessel wall. Under arterial flow conditions, platelets have an enhanced concentration near blood vessel walls. This non-uniform cell distribution depends on the fluid dynamics of blood as a heterogeneous medium. We use a parallelized lattice Boltzmann-immersed boundary method to solve the flow dynamics of red cells and platelets in a periodic 2D vessel with no-slip boundary conditions. Red cells are treated as biconcave immersed boundary objects with isotropic Skalak membrane tension and an internal viscosity five times that of the surrounding plasma. Using this method we analyze the influence of shear rate, hematocrit, and red cell membrane properties on lateral platelet motion. We find that the effective diffusion of platelets is significantly lower near the vessel wall compared to the center of the vessel. Insight gained from this work could lead to significant improvements to current models for platelet adhesion where the presence of red blood cells is neglected due to computational intensity.

5:46PM EE.00008 A Spatial-Temporal Model of Platelet Deposition and Blood Coagulation Under Flow , KARIN LEIDERMAN GREGG, AARON FOGELSON, University of Utah — In the event of a vascular injury, a blood clot will form to prevent bleeding. This response involves two intertwined processes: platelet aggregation and coagulation. Activated platelets are critical to coagulation in that they provide localized reactive surfaces on which many of the coagulation reactions occur. The final product from the coagulation cascade directly couples the coagulation system to platelet aggregation by acting as a strong activator of platelets and cleaving blood-borne fibrinogen into fibrin which then forms a mesh to help stabilize platelet aggregates. Together, the fibrin mesh and the platelet aggregates comprise a blood clot, which in some cases, can grow to occlusive diameters. Transport of coagulation proteins to and from the vicinity of the injury is controlled largely by the dynamics of the blood flow. It is crucial to learn how blood flow affects the growth of clots, and how the growing masses, in turn, feed back and affect the fluid motion. We have developed the first spatial-temporal model of platelet deposition and blood coagulation under flow that includes detailed decriptions of the coagulation biochemistry, chemical activation and deposition of blood platelets, as well as the two-way interaction between the fluid dynamics and the growing platelet mass.

5:59PM EE.00009 Enhancement of Absorption by Micro-Mixing induced by Villi Motion , YANXING WANG, JAMES BRASSEUR, GINO BANCO, Penn State Univ — Motions of surface villi create microscale flows that can couple with lumen-scale eddies to enhance absorption at the epithelium of the small intestine. Using a multigrid strategy within the lattice-Boltzmann framework, we model a macro-scale cavity flow with microscale "villi" in pendular motion on the lower surface and evaluate the couplings between macro and micro-scale fluid motions, scalar mixing, and uptake of passive scalar at the villi surface. We study the influences of pendular frequency, villous length, and villous groupings on absorption rate. The basic mechanism underlying the enhancement of absorption rate by a villous-induced "micro-mixing layer" (MML) is the microscale "pumping" of low concentration fluid from between groups of villi coupled with the return of high concentration fluid into the villi groups from the macroscale flow. The MML couples with the macrosacle eddies through a diffusion layer that separates micro and macro mixed layers. The absorption rate increases with frequency of villi oscillation due to enhanced vertical pumping. We discover a critical villus length above which absorption rate increases significantly. The absorption is influenced by villus groupings in a complex way due to the interference between vertical and horizontal geometry vs. MML scales. We conclude that optimized villi motility can enhance absorption and may underlie an explanation for the existence of villi in the gut. [Supported by NSF]

6:12PM EE.00010 Multiscale modeling of blood flow in cerebral malaria , DMITRY FEDOSOV, PhD student, Division of Applied Mathematics, Brown University, BRUCE CASWELL, Professor, Division of Engineering, Brown University, GEORGE KARNIADAKIS, Professor, Division of Applied Mathematics, Brown University — The main characteristics of the malaria disease are progressing changes in red blood cell (RBC) mechanical properties and geometry, and its cytoadhesion to the vascular endothelium. Malaria-infected RBCs become considerably stiff compared to healthy ones, and may bind to the vascular endothelium of arterioles and venules. This leads to a significant reduction of blood flow, and eventual vessel obstruction. Due to a non-trivial malaria-infected RBC adhesive dynamics and obstruction formations the blood flow in cerebral malaria is extremely complex. Here, we employ multiscale modeling to couple nanometer scales at the binding level, micrometer scales at the cell level and millimeter scales at the arteriole level. Blood flow in cerebral malaria is modeled using a coarse-grained RBC model developed in our group. The RBC adhesion is simulated based on the stochastic bond formation/breakage model, which is validated against recent experiments.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session EF Microfluidics: Devices III 101F

4:15PM EF.00001 Taylor-Aris Dispersion in Retentive, Ordered Pillar Array Columns[1] , XIAOHONG YAN, University of Pennsylvania and Xi'an Jiaotong University, QIUWANG WANG, Xi'an Jiaotong University, HAIM BAU, University of Pennsylvania — The method of volume averaging is applied to estimate the dispersion coefficient of solute advected in ordered pillar array columns with wall retention of the type used in chromatographic separation. The mass transfer resistance of the stationary phase is accounted for. The appropriate closure equations are solved in a unit cell to obtain the dispersion tensor as a function of the Peclet number, pillar pattern and size, and solute properties. The contributions of the flow and the wall adsorption to the dispersion are identified and discussed. The model is verified by comparing its predictions and obtaining favorable agreement with the results of direct numerical simulations and with available experimental data for columns containing pillars. The model is then used to optimize the pillars' shape and pattern to minimize longitudinal dispersion. It is demonstrated that by judicious selection of the pillars' shape and pattern, one can achieve significant reduction in the longitudinal dispersion coefficient.

[1]The work is supported by NSF STTR grant #0822723 to Vegrandis (HHB), China State Scholarship Fund (XY), and National Natural Science Foundation of China grant # 50806056 (QW).

4:28PM EF.00002 Axial Dispersion in Segmented Gas-Liquid Flow: Effects of the Channel Curvature[1] , METIN MURADOGLU, Koc University — The effects of channel curvature on the axial dispersion in segmented gas-liquid flows have been studied computationally in a two-dimensional setting using a front-tracking/finite-volume method. Passive tracer particles are used to visualize and quantify the axial dispersion. The molecular diffusion is modeled by random walk of tracer particles. It is found that there is significant axial dispersion in serpentine channels even in the absence of molecular diffusion and dispersion increases with channel curvature. It is known that there is no dispersion in straight channels since a lubricating thin liquid layer persists on the wall. However this lubricating liquid layer is periodically broken in the curved channel case leading to enhanced axial dispersion. It is found that the dispersion increases as the Peclet number (Pe) decreases both in straight and curved channels. Difference between the straight and curved channel decreases continuously as the Peclet number decreases and virtually disappears at low Peclet numbers, i.e., $Pe < 10$ in the present study. A model is proposed based on the difference between the liquid film thicknesses on the inner and outer side of the bend in the limit as $Pe \to \infty$. Good agreement is found between the computational results and the model when the liquid slug is well mixed by the chaotic advection.

[1]This work is suuported by Turkish Academy of Sciences (TUBA-GEBIP) and TUBITAK 108M238.

4:41PM EF.00003 A Micro-PIV Study of the Pulsed Micro-Flows Driven by an Insulin Pump , BING WANG, Aerospace Engineering Dept., Iowa State University, AYODEJI DEMUREN, Mechanical Engineering Dept., Old Dominion University, ERIC GYURICSKO, Children's Hospital of The King's Daughters, Eastern Virginia Medical School, HUI HU, Aerospace Engineering Dept., Iowa State University — In recent years, there is a surge in the popularity of using insulin pump or continuous subcutaneous insulin infusion therapy, as opposed to multiple daily injections by insulin syringe or an insulin pen. Some case studies have suggested that insulin delivery failure may be caused by precipitation of insulin within the infusion set. Speculation also exists that the flow of insulin through an insulin infusion set may be reduced or inhibited by air bubbles entrained into the micro-sized tubing system since there are chances that air be introduced into the insulin reservoir during the filling process. In the present study, a microscopic Particle Image Velocimtry (micro-PIV) system was used to characterize the transient behavior of the pulsed micro-flows inside the micro-sized tubing system of an insulin infusion set with insulin pump operating in basal mode (i.e., pulsed insulin pumping). The effects of the air bubbles entrained into the micro-sized tubing system on the insulin delivery process were assessed based on the micro-PIV measurements.

4:54PM EF.00004 A Methodology for Time-Resolved microDPIV , JAIME SCHMIEG, School of Biomedical Engineering and Sciences, Virginia Tech, ADRIC ECKSTEIN, Center for Advanced Aviation Systems Development, The MITRE Corporation, JOHN CHARONKO, PAVLOS VLACHOS, School of Biomedical Engineering and Sciences, Virginia Tech — Micro Digital Particle Image Velocimetry (uDPIV) measurements are often limited to time averaged analyses due to low signal to noise ratios, high background illumination, and low particle seeding. As a result, the measurement of transient microscale flows is difficult to achieve through conventional DPIV correlation methods. Eckstein and Vlachos (2009), presented the Robust Phase Correlation (RPC) method which utilizes a series of digital filters to mitigate the effects of background noise. This study further explores the potential of RPC using experimentally derived, time-resolved uPIV images taken within three different microchannels of various geometries. Performance comparisons were based on RMS error, as well as percent of erroneous vectors, as determined by the mode-ratio bootstrapping method (Pun et. al. 2007). Results displayed a significant reduction of RMS error and erroneous vectors for the RPC method in comparison to standard techniques. 1. Eckstein and Vlachos. Meas. Sci. Technol. (2009). 2. Pun et al. Meas. Sci. Technol. (2007).

5:07PM EF.00005 Inertial and channel confinement effects on laminar flow in microchannels with superhydrophobic surfaces , YONGPAN CHENG, CHIANGJUAY TEO, BOOCHEONG KHOO, National University of Singapore, LAB OF FLUID MECHANICS, DEPARTMENT OF MECHANICAL ENGINEERING, NATIONAL UNIVERSITY OF SINGAPORE TEAM — In order to reduce the pressure drop for flow through microchannels, the superhydrophobic surfaces which consist of micro-grooves, posts or holes are widely adopted. In this paper, the effective slip performances of transverse grooves, longitudinal grooves, posts and holes are investigated numerically. The numerical results show that the effective slip lengths of square posts, square holes and transverse grooves decrease with increasing Reynolds number, except those corresponding to longitudinal grooves. For small pattern width to channel height ratios, at low shear-free fractions, the effective slip length corresponding to square posts is equivalent of that of transverse grooves, and their slip lengths are lower than those of square holes and longitudinal grooves. With increasing shear-free fractions, the effective slip length of square posts surpasses those of square holes and longitudinal grooves. Square posts exhibit the highest effective slip length at extremely high shear-free fractions. This study may be useful for applications pertaining to the reduction of flow resistance in microchannels employing superhydrophobic surfaces.

5:20PM EF.00006 Designing ridged microchannels for continuous separation of cells based upon stiffness , ALEXANDER ALEXEEV, JOHN P. ARATA, Georgia Institute of Technology — The mechanical stiffness of human cells can be a key parameter that reveals the disease state of the cell, for example, in various cancers and in malaria. However, for stiffness to be utilized in diagnostic settings, we will require methods for continuous monitoring of the stiffness of cells in high throughput. Using three-dimensional computational modeling, we show that a pressure-driven microfluidic flow in a channel with solid diagonal ridges can effectively separate compliant microscopic particles, which represent biological cells and synthetic microcapsules. The flow separation is driven by an energy minimization process associated with periodical deformations of elastic particles in narrow constrictions. As a result of this process soft and stiff particles disperse to the opposite walls of microfluidic channel, thus enabling continuous flow separation of mechanically distinct particles. This simple microfluidic method could prove useful in various biomedical applications for continuous stiffness-based separation of biological cell.

5:33PM EF.00007 Interfacial rheology in two-phase microchannel flows , STEVEN HUDSON, JEFFREY MARTIN, NIST — Droplet dynamics are studied experimentally by particle tracer velocimetry and shape analysis to measure interfacial retardation in various flow fields, surfactant concentrations and interfacial sorption dynamics. Aqueous drops in mineral or silicone oils are investigated in Poiseuille flow in rectangular channels. The drop size (tens of microns) is small compared to the channel height and width. The drop size is also small compared to an intrinsic length scale defined by surfactant mass transport coefficients, so that the experiments may probe a regime where interfacial sorption dynamics are relevant. The interfacial mobility of the drop is compared with fully mobile predictions and is found to range from a few percent to nearly full mobility, as surfactant concentration is adjusted. The effect of channel asymmetry on drop circulation is also tested. The droplet shape relaxation rates are measured, indicating the local surfactant concentration and interfacial tension. The interfacial tension and Marangoni effects are thus measured in a single experiment.

5:46PM EF.00008 Separation and Stabilization of Deformable Drops in Microfluidics[1] , WINGKI LEE, Carnegie Mellon University, KATHARINA SCHRANK, RWTH Aachen University, LYNN WALKER, SHELLEY ANNA, Carnegie Mellon University — Microfluidic processes are effective for producing highly monodisperse droplet streams, but some desired processes inherently require the formation of polydisperse droplet populations. As an example, tipstreaming produces micron scale droplets along with larger drops that are 10-100 microns in size. Separation of these sizes is needed in order for the process to be useful. We have designed a microfluidic separator allowing fractionation of droplet sizes on-the-fly along with the injection of a surfactant-laden stream for further downstream stabilization of the fractionated emulsion. The behavior of the device is different for deformable emulsions than rigid particles, although the design can work for either. We report on the performance of the device as well as size distributions resulting from the fractionated populations.

[1]NSF CBET-0730727

5:59PM EF.00009 Pumpless femtoliter drop-on-demand generation via satellite formation[1] , DUSTIN MOON, DONG WOOK LEE, IN SEOK KANG, POSTECH — Manipulation of discrete or digital droplets can be a key process in chemical, pharmaceutical process in micro scale. This paper provides a drop-on-demand generation method of such micro droplets. Many drop generation techniques have been developed, from flow focusing to electrospray, dispensing droplets into different phases. But due to relatively long microchannel length and the use of syringe pumps, these methods show some limitations for single drop generation at a wanted time. Satellites, an unwanted droplet generated from liquid bridge breakup, have rather stable size distribution resulting from its rapid formation mechanism. In this experimental work, satellites are formed between metallic capillaries of outer diameter 910, 460, 260 micron respectively, and inside a PDMS channel of 100 micron-depth. To form a liquid bridge and break it, capillaries can be moved, or pulsed electric field is applied to deform, elongate droplets, and to form an unstable liquid bridge, and break. Several fluids from DI water to PEG with and without polystyrene particles and E.Coli. were used to form satellite drops in air and in oil. Details of each behavior were captured using Photron PCI 1024X high speed camera and analyzed accordingly.

[1]This research was funded by Center for Ultramicrochemical Process Systems sponsored by KOSEF.

6:12PM EF.00010 Droplet Dynamics in Two Phase Microflows , BRIAN CARROLL, CARLOS HIDROVO, The University of Texas at Austin — An experimental investigation is presented that addresses the engineering challenges of a two-phase, inertial-based micromixer. Results indicate a predictable pattern between the Reynolds number in the microchannel and the detached droplet size and geometric features, with increasing gas velocities leading to a high aspect ratio (elongated) detached slugs. Full droplet detachment and entrainment into the gaseous flow, where the droplet shares no interface with the solid wall, was not observed in the range of parameters tested. The influence of the solid wall during the collision and mixing process remains unresolved, although it is evident that increasing the surface energy of the solid boundary facilitates the coalescence and mixing process. Using Laser Induced Flourescence, the extent of mixing is quantified experimentally and compared for different detached droplet sizes, velocities, and microchannel geometries. This data, together with the conditions required for detachment and entrainment, will provide a more complete picture of an inertial-based, droplet collision micromixer.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session EG Biofluids IV: Cellular I 101G

4:15PM EG.00001 Complex motions of vesicles and capsules in flow , PETIA VLAHOVSKA, Dartmouth College, YUAN-NAN YOUNG, NJIT, CHAOUQI MISBAH, UJF-Grenoble, France — Membrane-bound particles exhibit rich dynamics when placed in flow. For example, in simple shear flow, vesicles made of lipid bilayers tank-tread or tumble. Capsules and red blood cells also show oscillations in the tank-treading inclination angle, called swinging. This motion originates from membrane shear–elasticity and non–spherical unstressed shape. We develop an analytical theory that quantitatively describes the swinging dynamics. Our analysis takes into account that the membrane is deformable, incompressible, and resists bending and shearing. Analytical results for the shape evolution are derived by considering a nearly-spherical particle shape. The phase diagram is constructed and compared to previous models which assume fixed ellipsoidal shape. Dynamics in quadratic and time-dependent flows is also discussed. Floquet analysis is conducted to investigate the vesicle dynamics and conditions for chaotic shape and flow dynamics are established.

4:28PM EG.00002 Why do red blood cells have asymmetric shapes even in a symmetric flow? , BADR KAOUI, Technische Universiteit Eindhoven (Eindhoven, The Netherlands) and CNRS - Universite de Grenoble I (Grenoble, France), GEORGE BIROS, Georgia Institute of Technology (Atlanta, GA, USA), CHAOUQI MISBAH, CNRS - Universite de Grenoble I (Grenoble, France) — Understanding why red blood cells (RBCs) move with an asymmetric shape (slipper-like shape) in small blood vessels is a longstanding puzzle of the blood circulatory research. We discover, by considering a phospholipid bilayer model (a biomimetic system of RBCs), that slipper results from the loss of stability of the symmetric shape. It is shown that the birth of slipper results in a significant decline of velocity difference between the cell and the imposed flow, providing thus higher flow efficiency for RBCs. An increase of membrane rigidity is found to lead to a dramatic change of the slipper morphology, offering thus a potential diagnostic for cell pathologies.

4:41PM EG.00003 Tank-treading, swinging, and tumbling of elastic capsules[1] , PROSENJIT BAGCHI, R. MURTHY KALLURI, Rutgers University — It is well known that deformable capsules, vesicles, and red blood cells undergo tank-treading or tumbling motion when freely suspended in shear flows. Recent experiments have shown the existence of a trembling or vacillating-breathing mode as well. Here we present three-dimensional simulations on the dynamics of elastic capsules in shear flow. Our objective is to investigate these different regimes of capsule dynamics, and the coupling between the shape deformation and orientation dynamics. By progressively increasing the viscosity ratio of the internal-to-external fluids at low shear rates, we predict the three distinct modes of motion: a swinging or oscillatory (OS) mode, a vacillating-breathing (VB) mode, and a pure tumbling mode (TU). We show how the coupling between the shape deformation and orientation dynamics influences the transition from the OS to VB to TU modes. At higher shear rates, we predict three new classes of transient motions during which the capsule switches from one mode to the other over time as (i) VB to OS, (ii) TU to VB to OS, and (iii) TU to VB. Phase diagrams showing various regimes are presented.

[1]Funded by NSF (BES-0603035, CTS-0625936)

4:54PM EG.00004 Simulations of the tumbling and tank treading motions of cells immersed in fluid flow , KENG-HWEE CHIAM, DUC VINH LE, A*STAR Institute of High Performance Computing — We describe the development of computational techniques to study the deformation of cells modeled as liquid capsules enclosed by thin shells suspended in fluid flow. Computations are performed for cells with spherical, ellipsoidal and biconcave unstressed shapes over a wide range of the dimensionless shear rate and for a broad range of the ratio of the internal to surrounding fluid viscosities. Results for small deformations of initially spherical capsules are in quantitative agreement with analytic predictions. Under large deformations, the results show that spherical capsules deform to stationary configurations and the membranes undergo steady tank-treading motion. For capsules of ellipsoidal and biconcave unstressed shapes, the capsules exhibit either tumbling, tank-treading of the membrane about the viscous interior with periodic oscillations of the orientation angle, or intermittent behaviour in which the two modes occur alternately depending on the viscosity, membrane elasticity and shear rate. Our method provides an efficient way to study the tank-treading-to-tumbling transition of red blood cells in shear flows as the shear rate decreases. Observations of such motions may provide a sensitive mean of assessing cell membrane properties. Finally, we also describe simulations of the long-time behavior of a dense suspension of red-blood cells in a micro-channel to illustrate the efficiency of the method.

5:07PM EG.00005 ABSTRACT WITHDRAWN –

5:20PM EG.00006 Dynamics of suspensions of elastic capsules flowing in confined geometries , PRATIK PRANAY, PIETER J.A. JANSEEN, MICHAEL GRAHAM, UW-Madison — Modeling the behavior of fluid-filled capsules (a simple representation of red blood cells and vesicles), is not only important to understand biological processes, such as blood flow in the microcirculation, but also to help design and improve microfluidic devices for characterizing or separating such particles. The present work describes simulations of large numbers of deformable capsules with various properties in confined geometries. Our algorithm incorporates a General-Geometry-Ewald-Like method (GGEM) for efficiently calculating hydrodynamic interactions (O(N)) in an immersed-boundary method. With our algorithm, we have addressed several issues. The ability to quickly simulate large number of particles enables examinations of not only of the competition between shear-induced diffusion and wall-induced hydrodynamic migration of single particles, but also exploration of concentration effects and segregation by size, shape and/or deformability. Combined with the simulation of grooved channels, we propose a methodology to separate these cells depending on their deformability and size. Finally, the effect of addition of long-chained polymer molecules in blood flow, known to lower blood pressure, is investigated.

5:33PM EG.00007 Numerical simulation of platelet margination in microcirculation , HONG ZHAO, ERIC SHAQFEH, Stanford University — The adhesion of platelets to vascular walls is the first step in clotting. This process critically depends on the preferential concentration of platelets near walls. The presence of red blood cells, which are the predominant blood constituents, is known to affect the steady state platelet concentration and the dynamic platelet margination, but the underlying mechanism is not well understood to-day. We use a direct numerical simulation to study the platelet margination process, with particular emphasis on the Stokesian hydrodynamic interactions among red cells, platelets, and vessel walls. Well-known mechanical models are used for the shearing and bending stiffness of red cell membranes, and the stiffer platelets are modeled as rigid discoids. A boundary integral formulation is used to solve the flow field, where the numerical solution procedure is accelerated by a parallel $O(N \log N)$ smooth particle-mesh Ewald method. The effects of red cell hematocrit and deformability will be discussed.

5:46PM EG.00008 The motion of a single red blood cell in a capillary , THIERRY SAVIN, L. MAHADEVAN, Harvard University — The collective vaso-occlusive event in sickle cell disease induced by multiple red blood cells (RBC's) has recently been evoked and controlled in vitro using a microfluidic platform [1]. The increase in the cells' stiffness in this disease is believed to be a predominant factor at the onset of the occlusion. We report here the motion of a single swollen RBC in a capillary. We use a tapered glass capillary with inner diameter as low as 3 microns, and track the squeezed cell driven by a controlled pressure drop. This allows us to simultaneously measure the variations of the RBC velocity as a function of the pressure gradient and of the local capillary diameter in a single experiment. We show that under certain regimes of confinement, the velocity increases with the pressure head with a characteristic power-law. We analyze our findings in terms of a elasto-hydrodynamical model for soft lubrication.

[1] Higgins et al., Proc. Natl. Acad. Sci. U.S.A. **104**: 20496 (2007).

5:59PM EG.00009 Simulation of red blood cells flowing over wall-bound cells , AMIR H.G. ISFAHANI, University of Illinois at Urbana-Champaign, HONG ZHAO, Stanford University, JONATHAN B. FREUND, University of Illinois at Urbana-Champaign — Intercellular dynamics play a critical role in the phenomenology of the microcirculation. We present a quantitative investigation of the forces exerted by red cells on protrusions on a microvessel of diameter around 12 μm, which is 1.5 times the longest dimension of a red cell at rest. This configuration serves as a model for white blood cells (leukocytes), which can bind nearly statically to the endothelial cells as part of the inflammation response. The simulation tools are based on an $O(N \log N)$ boundary integral formulation. It permits the cells to both be realistically flexible and to approach to very close separation distances. The red blood cells are modeled as finite-deformation elastic membranes with strong resistance to surface dilatation and relatively small but finite resistance to bending. The no-slip condition is applied on the protrusion as well as the vessel walls. Simulation results show that these forces are significantly augmented by the particulate character of blood. For a tube hematocrit of 30% and a hemispherical protrusion with a height to tube diameter ratio of 0.4, the average forces are increased by about 50% and the local forces by more than two folds relative to forces from an effective viscosity homogenized blood. Different flow configurations are considered and analyzed.

6:12PM EG.00010 A computational study of male pronuclear migration in the C. elegans embryo , TAMAR SHINAR, Courant Institute, New York University, FABIO PIANO, Center for Genomics and Systems Biology and Department of Biology, New York University, MICHAEL SHELLEY, Courant Insitute, New York University — After fertilization the one-celled C. elegans embryo undergoes a series of complex but stereotyped dynamics that lead to proper progression of early development. This system offers a great opportunity to combine modeling and experimental approaches to learn about the biophysical properties underlying fundamental developmental events. In particular, we study the mechanisms underlying male pronuclear migration using a detailed computational model that captures important features of the system. We model the cytoplasmic flow as a Stokes fluid, accounting for the enclosing cell geometry. The fluid is two-way coupled to a rigid pronucleus that is subject to forces computed based on the dynamic instability model of microtubule dynamics. We use the computational model to study force models for microtubule based motility as well as the effects of the fluid drag and geometric confinement on the pronucleus and microtubules.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session EH Drops III: Collisions and Coalescence 101H

4:15PM EH.00001 Coalescence of Low-Viscosity Liquids , J.D. PAULSEN, S.R. NAGEL, James Franck Institute, University of Chicago — When two fluid drops come into contact, a dramatic topological transformation occurs as the drops coalesce. The speed and geometry of this finite-time singularity make it difficult to study optically, so we use an electrical method to probe coalescence at early times.[1,2] For low-viscosity liquids, we measure a resistance that varies as t^{-1} at early times and as $t^{-1/2}$ at late times. In the inviscid case, these power laws have been interpreted with a model in which the drops coalesce at a slightly deformed interface.[2] In order to test further predictions of this model, we study coalescence at faster rates than previously attainable. We have previously shown that the crossover time between these power laws increases with viscosity, depends weakly on the surrounding gas pressure, but does not depend on the weight of the gas.[3] We further explore outer-fluid effects by replacing the ambient gas with a liquid, showing that the crossover time is delayed with increasing outer-fluid viscosity.

[1]J. C. Burton, J. E. Rutledge, and P. Taborek, Phys. Rev. Lett. **92**, 244505 (2004).
[2]S. C. Case and S. R. Nagel, Phys. Rev. Lett. **100**, 084503 (2008).
[3]J. D. Paulsen, J. C. Burton, and S. R. Nagel, DCMP 2009.

4:28PM EH.00002 The effect of interfacial slip on the drainage time to coalescence between two droplets , L. GARY LEAL, KOSTAS TSIGLIFIS, ARUN RAMACHANDRAN, ANSHUMAN ROY, University of California, Santa Barbara — A fundamental question that arises in the coalescence of two drops in a flow is the dependence of the drainage time t_d prior to film rupture on the capillary number *Ca* and the dimensionless Hamaker constant, which is inversely proportional to the square of the drop radius, R^{-2}. Prior investigations from our group have shown that predictions of this relationship based on both scaling theory and numerical simulations deviate qualitatively from experimental data. We believe that a possible explanation for these discrepancies is a breakdown of the continuum flow model for the extremely thin films that are realized prior to film rupture. Such a breakdown would most likely first manifest itself as a violation of the no-slip condition at fluid interfaces. In this work, we examine the effect of interfacial slip on the dependence of the drainage time with capillary number for different λ via boundary integral simulations. Interfacial slip is modeled via the Navier-slip condition, and the slip parameter employed in the simulations is predicted using the work of Goveas and Frederickson [*Eur. Phys. J. B* **2**, 79–92 (1998)]. The agreement with the scaling exponents of t_d versus *Ca* and$R$ is improved, but the absolute values of the drainage times are lower than the experimental values. Possible reasons for these deviations are explored.

4:41PM EH.00003 A critical angle for electrocoalescence of conical droplets , JAMES BIRD, Harvard University, WILLIAM RISTENPART, University of California at Davis, ANDREW BELMONTE, Pennsylvania State University, HOWARD STONE, Princeton University — Oppositely charged droplets suspended in air attract one another and, when the droplets are sufficiently close, electrical stresses deform the leading edges into cones. Here we investigate whether or not the liquid cones coalesce immediately following contact. Using high-speed imaging, we find that the coalescence behavior depends on the cone angle, which we control by varying the drop size and the applied voltage across the drops. The two drops coalesce when the slopes of the cones are small, but recoil when the slopes exceed a critical value. We propose a surface energy model (volume-constrained area minimization) to describe the transition between these two responses. The model predicts a critical cone intercept angle of 30.8°, which is in good agreement with our measurements.

4:54PM EH.00004 Deformation and merging of droplets at different electric potentials[1] , DONG WOOK LEE, IN SEOK KANG, POSTECH — In the present work, two droplets which are attached to different electrodes are faced each other and then, approaching each other very slowly, deformation and merging of the two droplet were closely watched using high-speed camera. It was found from the observation that this phenomenon can be separated into three phases; deformation, liquid bridge formation, merging. First, in the deformation phase surface tension and electric force achieve the equilibrium and the deformed shape is stable. Second, in the liquid bridge formation phase electric force is much bigger than surface tension and the liquid bridge is made very quickly between the tips of droplets. Lastly, in the merging phase the two droplets are merging because of surface tension. We focused the minimum distance to make liquid bridge and to merge the two droplets under constant potential difference. Finally, we performed numerical simulation using level set method and compared the experimental result with the numerical result.

[1] This work was supported by the grants from Korean Science and Engineering Foundation (KOSEF) and the BK21 program of the Ministry of Education, Science and Technology (MEST), Korea. The authors acknowledge greatly the financial supports.

5:07PM EH.00005 Non-coalescence of oppositely charged drops , W.D. RISTENPART, Univ. Calif. at Davis, J.C. BIRD, Harvard Univ., A. BELMONTE, Penn. State Univ., F. DOLLAR, H.A. STONE, Harvard Univ. — Electrically induced droplet motion manifests itself in processes as diverse as storm cloud formation, commercial ink-jet printing, petroleum and vegetable oil dehydration, electrospray ionization in mass spectrometry, electrowetting and lab-on-a-chip manipulations. An important issue in practical applications is the tendency for adjacent drops to coalesce, and oppositely charged drops have long been assumed to experience an attractive force that favors their coalescence. Here we report the existence of a critical field strength above which oppositely charged drops do not coalesce. We observe that appropriately positioned and oppositely charged drops migrate towards one another in an applied electric field; but whereas the drops coalesce as expected at low field strengths, they are repelled from one another after contact at higher field strengths. Qualitatively, the drops appear to "bounce" off one another. We directly image the transient formation of a meniscus bridge between the bouncing drops, and propose that this temporary bridge is unstable with respect to capillary pressure when it forms in an electric field exceeding a critical strength. The observation of oppositely charged drops bouncing in strong electric fields should affect our understanding of any process involving charged liquid drops, including de-emulsification, electrospray ionization and atmospheric conduction.

5:20PM EH.00006 Partial Coalescence of Oppositely Charged Drops , J.C. CREASEY, B.S. HAMLIN, W.D. RISTENPART, Univ. Calif. at Davis — Oppositely charged drops fail to coalesce above a critical field strength, despite the attractive force between the opposite charges [1]. Here we investigate the coalescence behavior at intermediate field strengths for charged water drops in oil, and we report that under many conditions the droplets undergo partial coalescence, i.e., a smaller daughter droplet is expelled. This partial coalescence is highly sensitive to the ionic strength of the droplets. For a given field strength, there exists a critical ionic strength above which the drops completely fail to coalesce and below which they partially coalesce. We explore the roles of charge density, drop size, inertia and viscous drag on the partial coalescence behavior and we interpret the results in terms of a competition between the respective time scales for hydrodynamic motion and ionic conduction. [1] Ristenpart, Bird, Belmonte, Dollar & Stone, *Nature*, in press (2009).

5:33PM EH.00007 A New Approach to Modeling Drop-Pair Collisions: Predicting the Outcome through a Fluidic-Mechanical System Analogy , PAUL VAN NOORDT, CARLOS HIDROVO, The University of Texas at Austin — The study of microfluidics has proven to be of great value in many engineering and scientific applications. Because of the small scales involved, microfluidics requires only small sample sizes, which can result in shorter reaction and analysis times, relatively cheap costs, and little waste. In this study, we investigate the process of two drops colliding head-on in order to gain a better understanding of the mechanisms that govern the outcome of the collision. The relationship between kinetic and surface energy of the colliding drops is considered, as is the viscosity of the intervening gaseous medium, as factors that govern the outcome. The collision process is modeled by a squeeze-flow problem involving both planar and non-planar geometry, with attention given to the deformation of the interacting surfaces. Based on the nature of the collision process, an analogy is made between the fluidic systems of colliding liquid bodies and a mechanical mass-spring-damper system. Examination of the analogous mechanical system yields the derivation of an effective damping ratio, ζ^*, which is used to predict the outcome of the drop-drop collision. Predictions made by utilizing the effective damping ratio are then compared to numerical results and experimental data found in the literature.

5:46PM EH.00008 Characteristics of Unequal Size Drop Collisions[1] , JUNGYONG KIM, Samsung Engineering Co. Ltd., ELLEN LONGMIRE, University of Minnesota, MAN SIK KIM, Samsung Engineering Co. Ltd. — Pairs of water/glycerin drops were injected into silicone oil and traveled on downward trajectories before colliding. Unequal size drop collisions with drop size ratios (D_s/D_L) of 0.7 and 0.5 were investigated. Simultaneous dual-field PIV measurements were obtained to characterize coalescence and rebounding behavior. The initial injection angle and tube height were adjusted to access appropriate impact parameters. In the current study, the collision angle of the large drop was, in general, shallower than that of the small drop, and a range of velocity ratios and impact parameters was examined. Coalescence occurs above *We** = 11 similar to collision outcomes for equal size drops. As drop size ratio decreases, the intervening film deforms more. If the velocity ratio $u_L/u_s \quad < 1$, the interface remains deformed at coalescence, but if $u_L/u_s \quad > 1$, the interface flattens before coalescence. The rupture location varies due to the asymmetry of the drops. As collision offset increases (B $>$ 0), the film rupture time is shortened and mixing of the fluid within the drops is enhanced after coalescence. These results will be compared with the behavior observed previously for equal size drop collisions.

[1]Supported by Petroleum Research Fund (42939-AC9), NSF (CTS-0320327), and Samsung Engineering, Co. Ltd.

5:59PM EH.00009 Bifurcation and Stability of a System of n Coupled Droplet Oscillators with S_n symmetry , DAVID SLATER, PAUL STEEN, Cornell University — The dynamics of a large array of interacting droplets is of interest in a variety of applications and, on its own, as a nonlinear dynamical system. A network of n spherical-cap droplet oscillators are coupled via a central reservoir such that the system has S_n symmetry. Under a constant-volume constraint, the inviscid case is modeled as a system of $n-1$ second order differential equations. Surface tension resists the inertia of deformations from the spherical shape. The symmetry of the system is important. In particular, independent of the equations, equilibrium solutions can be categorized by symmetry group into families, each with some p large and some $q = n-p$ small droplets. Within each family stability is invariant, which greatly reduces the number of cases to consider. Equilibrium curves and their stability are calculated analytically for an arbitrary number of droplets in the preferred coordinate space. For small volumes, the only equilibrium state is stable and corresponds to all identical droplets. For larger volumes, a multitude of equilibrium states exist, each having the property that all droplets have equal radius of curvature. Nearly all these equilibrium are unstable, the only stable configuration being one droplet large and the rest small. The nonlinear dynamics of the three droplet case is examined numerically and exhibits quasiperiodic, periodic and chaotic dynamics.

6:12PM EH.00010 Dynamics of bubbles and drops in a Hele-Shaw cell , KO OKUMURA, AYAKO ERI, MARIA YOKOTA, Department of Physics, Ochanomizu University — Bubbles created in liquid and drops moving in another immiscible liquid are easy to be observed from the side when enclosed in thin space made by two parallel plates, i.e. in a Hele-Shaw cell, and the results thus obtained should be interesting to be compared with three dimensional counterparts to find dimensional crossover. We show two such experimental examples: (1) thinning dynamics of liquid film encapsulating an air bubble and (2) coalescence dynamics of a liquid drop to the bath phase of the same liquid. Our experimental results are well explained by simple theories, providing the physical understanding of the phenomena.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session EJ Bubbles III 101I

4:15PM EJ.00001 Solid object impact creates supersonic air flow , STEPHAN GEKLE, IVO PETERS, Physics of Fluids, U Twente, The Netherlands, JOSE MANUEL GORDILLO, Universidad de Sevilla, Spain, DEVARAJ VAN DER MEER, DETLEF LOHSE, Physics of Fluids, U Twente, The Netherlands — When an object impacts on a water surface a sizeable cavity is formed below the surface which subsequently collapses due to hydrostatic pressure. The surrounding air is first sucked into the expanding cavity while at a later stage it is squeezed out of the - then shrinking - cavity. We measure this air flow using high-speed imaging of fine smoke particles. Numerical simulations combining a boundary-integral method with a fully compressible Euler solver reveal the intricate structure of the gas dynamics. Despite an impact velocity of merely 1 m/s and a cavity overpressure of only 0.02 atmospheres the air flow is shown to attain supersonic speeds. Consequently, there are significant effects of this air flow close to pinch-off which can be observed consistently in experiment and numerics: (i) the cavity wall is not smoothly curved but exhibits a kink and (ii) the cavity neck is pushed upwards.

4:28PM EJ.00002 The water entry of decelerating spheres , JEFFREY ARISTOFF, Princeton University, TADD TRUSCOTT, Naval Undersea Warfare Center Newport, ALEXANDRA TECHET, JOHN BUSH, Massachusetts Institute of Technology — We present the results of a combined experimental and theoretical investigation of the vertical impact of low-density spheres on a water surface. Particular attention is given to characterizing the sphere dynamics and the influence of its deceleration on the shape of the resulting air cavity. A theoretical model is developed that yields simple expressions for the pinch-off time and depth. Theoretical predictions compare favorably with our experimental observations, and allow us to rationalize the form of water-entry cavities resulting from the impact of buoyant and nearly buoyant spheres.

4:41PM EJ.00003 Cavitation induced Particle Motion on Surfaces: A Model for Particulate Contamination removal[1] , S. ROBERTO GONZALEZ-A, XIAOHU HUANG, PEDRO QUINTO-SU, TOM WU, CLAUS D. OHL, Nanyang Technological U — The motion of adherent polystyrene particles accelerated with a transient cavitation bubble is investigated experimentally. The bubble and particle dynamics is recorded with a high-speed camera. The particle trajectory is studied as a function of the initial separation and the particle diameter. For all particles investigated, i.e. 10, 4.5 and 2 μm in diameter, we find the same intriguing dynamics: a short initial separation (<0.7 Rmax) leads to a final displacement away from the bubble while particles at larger separations are effectively attracted towards the bubble. All particles follow the same master curve when the initial separation is nondimensionalized with the bubble diameter and the final displacement with the particle diameter. Our comparison of the trajectory to a force balance model indicates that unsteady boundary layers have to be taken into account. We studied the rolling dynamics induced by the strong shear flow and find angular speeds in excess of 400,000 rps for very brief times. We find that both torque and drag are important mechanisms for the removal of particulate contamination in cavitation based cleaning methods.

[1]The authors acknowledge the funding through the Ministry of Education, Singapore (T208A1238).

4:54PM EJ.00004 Numerical analysis on cavitation inception in the rotary valve of the power steering system[1] , GWANG NYEON RYU, Department of Mechanical and Aerospace Engineering, Seoul National University, SUN HONG PARK, MANDO, MYUNG HWAN CHO, JUNG YUL YOO, Department of Mechanical and Aerospace Engineering, Seoul National University — The power steering valve directs the power steering oil to either side of a power piston and relieves the driver of the effort to turn the wheel, when a driver begins to operate the vehicle. It is well known that hiss noise occurring at that moment is caused mainly by cavitation of the oil inside the rotary valve. However, it is very difficult to check the flow pattern and the cavitation inception experimentally because of the complex geometry and high pressure in the interior of the rotary valve. In spite of the increasing interest in this flow phenomenon inside the rotary valve, only few studies have been reported. In the present study, this complex oil flow inside the rotary valve has been analyzed numerically using three-dimensional cavitation model provided by the commercial code, FLUENT 6.2. It is confirmed that the location of cavitation inception is similar to that obtained by the existing two-dimensional numerical analysis. Then, the volume fraction of oil vapor has been compared with the hiss noise level measured experimentally in a semi-anechoic room.

[1]Supported by MANDO.

5:07PM EJ.00005 Probing nanotubes and red blood cells with laser-induced cavitation bubbles , PEDRO QUINTO-SU, XIAOHU HUANG, CLAUDIA KUSS, ROBERTO GONZALEZ, PETER PREISER, TOM WU, CLAUS-DIETER OHL — A spatial light modulator (SLM) is used to simultaneously create arrays of laser-induced cavitation bubbles. The different bubble geometries result in the creation of a directed, transient and strong liquid flow. Due to the fast dynamics of the cavitation bubbles the flow is actuated on very short temporal (μs) and spatial (μm) scales. We show two examples of the use of laser-induced cavitation bubbles to probe and manipulate small objects: multiwalled carbon nanotubes (MWCNT) and red blood cells (RBCs). In the case of MWCNT, we use a pair of bubbles to displace and bend the nanotubes. By measuring the time it takes for the nanotube to recover its original shape we can estimate the flexural rigidity and the bending modulus. The shape recovery is recorded with a high-speed camera at up to 300,000 frames per second (fps). We found the flexural rigidity to be on the range of 0.98 – 6.6 $\times$ 10^{-19} Nm^2 and the Young's modulus on the order of 0.06-0.6 TPa for MWCNT with an average diameter of 117.8 $\pm$ 6.7 nm and a thickness of 4.6 $\pm$ 0.75 nm. A similar approach is used to study the mechanical properties of RBC's, where multiple cells are elongated due to the radial flow induced by a single bubble. We study the shape recovery of the RBCs and find a significant difference when they are treated with an enzyme.

5:20PM EJ.00006 Bubble Density Functionals and Stochastic Fluid Flow Models , RAY B. STOUT, RhoBetaSigmaAffaires, Livermore, CA 94450 — Arbitrary spatial domains containing a stochastic density of gas pressurized bubbles in a fluid material are statistically decomposed such that the probable relative position between two arbitrary spatial points is a time dependent, but path independent, integral functional of fluid material and the bubble density function. Since the relative position vector is an invariant physical quantity, the line integral is stochastically path independent. Using this spatial decomposition vector, arbitrary spatial domains for probable fluid and probable bubble areas and volumes are derived. To address instability in bubble density dependent fluid flows, bubble density evolution in fluids is then formulated with: (1). A Boltzmann bubble density field equation to describe discrete bubble species size(radius) and gas content(atoms) evolution; (2). Relative deformation and velocity functionals of bubble density; (3). Fluid material and bubble gas mass transport functionals of bubble density; and (4). Momentum transport and stress/bubble-pressure functionals of bubble density. These stochastic equations embed finite physical length scales of bubble density fluids.

5:33PM EJ.00007 Numerical and experimental study of the coefficient of restitution for colliding bubbles , ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico, DOMINIQUE LEGENDRE, Institut de Mecanique des Fluides de Toulouse — We have studied the motion of bubbles colliding with solid walls both numerically and experimentally. The simulations were performed considering a VOF method of the JADIM code (Bonometti & Mahnaudet IJMF 2007) that permits to reproduce accurately motion bubbles for a large range of Eotvos and Morton numbers. To be able to compare with the numerical results, the experiments were carried out using silicon oils, for which the interface remains clean under ordinary laboratory conditions. By measuring the approach and rebound velocities, we calculated the coefficient of restitution of the collision, ϵ, which was found to scale as $-\log \epsilon \sim (Ca/St^*)^{1/2}$, as suggested by Zenit and Legendre (PoF, 2009) (where Ca is the capillary number and St^* is a modified Stokes number). Since the numerical results were validated (through direct comparisons with experiments), we conducted a vast parametric study of the coefficient of restitution, varying all the fluid properties in an independent manner. We will discuss these results and their implications in the study of solid/fluid particle collisions in general.

5:46PM EJ.00008 Enhanced slip on micro-patterned substrates due to contact line depinning , JAMES FENG, PENG GAO, Department of Chemical and Biological Engineering, University of British Columbia — We present numerical simulations of a shear flow over a periodically patterned substrate with entrapped gas bubbles. A diffuse-interface model is employed to handle the liquid–gas interface deformation and the three-phase contact line. Depending on the shear rate and the pattern geometry, four flow regimes are observed. The contact lines can be pinned, depinned or eliminated depending on the competition between the shear force and the surface tension. The effective slip length is found to be dependent on the morphology of the menisci and hence on the shear rate. In particular, the bubbles are transformed into a continuous gas film when the shear rate is larger than a critical value, resulting in a significantly enhanced slip length proportional to the liquid–gas viscosity ratio. The present results have interesting implications for effective slip on superhydrophobic surfaces.

5:59PM EJ.00009 Generation of mysterious bubbles by irradiation of femtosecond pulses in ultrapure water , TAKAYUKI SAITO, MANABU YAMAMOTO, Shizuoka University, SHINGO OISHI, SHIN-ICHIRO AOSHIMA, Hamamatsu Photonics K.K. — The femtosecond pulse laser (fs pulse), which is a high-intensity and ultrashort light pulses, produces the optical nonlinear phenomenon such as multiphoton absorption, and induces the completely new phenomenon which is not obtained by using conventional laser pulses. In this research, fs pulses were irradiated into pure water, and micro bubbles were generated. We conducted time-resolved measurement in order to investigate the process of the bubble generation in detail by using the pump probe method. As a result, it was observed that strong light was emitted in the middle of the region of refractive index changed in the water at 900 ps, and subsequently, a bubble was generated with about 1 ns after the fs pulse irradiation into the water.

6:12PM EJ.00010 Enhanced Boiling Heat Transfer using Acoustic Interfacial Actuation[1] , THOMAS R. BOZIUK, MARC K. SMITH, ARI GLEZER, Georgia Institute of Technology — Low-power acoustic actuation is used to enhance boiling heat transfer on a submerged surface and inhibit the instabilities that lead to film boiling at the critical heat flux by controlling the formation and evolution of the vapor bubbles. The effects of the acoustic field are investigated using a flat boiling heat transfer surface having a central, isolated hot spot that is designed to control the location, growth, and detachment of a single vapor bubble using a thin hydrophobic coating centered around the hot spot. Specific emphasis is placed on the coupling between the frequency and sound pressure and bubble diameter. It is shown that the acoustic field induces interfacial instabilities that affect the bubbles' contact line with the surface leading to their detachment. In addition to contact line dynamics, the primary and secondary Bjerknes forces play an important role in the detachment and advection of vapor bubbles.

[1]Supported by ONR.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session EK Particle Laden Flows I: Structure and Coating 101J

4:15PM EK.00001 Modeling Particle Concentration In Slurry Flows Using Shear-Induced Migration: Theory vs. Experiments , KANHUI LIN, Hong Kong University, PAUL LATTERMAN, University of California, Los Angeles, TRYSTAN KOCH, Harvey Mudd College, VINCENT HU, JOYCE HO, MATTHEW MATA, NEBOJSA MURISIC, ANDREA BERTOZZI, University of California, Los Angeles — Different flow regimes observed in our experimental study of particle-laden thin film flows are characterized by differing particle concentration profiles. We develop a theoretical model for particle concentration in order to capture our experimental observations. Our model is based on equilibrium assumption and it incorporates all relevant physical mechanisms, including shear-induced particle migration and settling due to gravity. It leads to a coupled system of ordinary differential equations for particle volume fraction and shear, which are solved numerically for various parameter sets. We find excellent agreement between our numerical results and experimental data. Our model is not only successful in reproducing the experimentally observed regimes, but also in capturing the connection between these regimes and the experimental parameters.

4:28PM EK.00002 Slurries: An Experimental Study Of Gravity-Driven Particle-Laden Thin Film Flows , JOYCE HO, VINCENT HU, University of California, Los Angeles, TRYSTAN KOCH, Harvey Mudd College, PAUL LATTERMAN, University of California, Los Angeles, KANHUI LIN, Hong Kong University, MATTHEW MATA, NEBOJSA MURISIC, ANDREA BERTOZZI, University of California, Los Angeles — An experimental study of gravity driven particle-laden thin film flows reveals several different regimes: particles either settle onto the solid substrate and out of the flow, they accumulate in the contact line region, or remain well mixed throughout the liquid layer. We carry out extensive experiments using liquids with varying viscosity, several different particle sizes and a wide range of particle concentrations and inclination angles, and accurately determine which regime characterizes each considered configuration. We compile our experimental observations and construct phase diagrams which clearly indicate the influence of the experimental parameters on the observed flow regime. In particular, our results reveal both an interesting influence of particle size on mixture dynamics as well as a connection between observed flow regime and the development of fingering instability at the contact line of a particle- laden thin film.

4:41PM EK.00003 An eXtended Finite Element Method (XFEM) for the simulation of the flow of viscoelastic fluids with suspended particles , YOUNG JOON CHOI, MARTIEN HULSEN, HAN MEIJER, Eindhoven University of Technology — We present an eXtended Finite Element Method (XFEM) combined with DEVSS-G/SUPG formulations for the direct numerical simulation of the flow of viscoelastic fluids with suspended rigid particles. In this method, the finite element shape functions are extended through the partition of unity method (PUM) by using virtual degrees of freedom as the enrichment for the description of discontinuities across interface. For the whole computational domain including both the fluid and particles, we use a regular mesh which is not boundary-fitted. Then, the fluid domain and the particle domain are fully decoupled by using the XFEM enrichment procedures. The no-slip boundary condition on the interface between fluid and rigid body is realized by using constraints implemented with Lagrange multipliers. For moving particle problems, we incorporate a temporal arbitrary Lagrangian-Eulerian (ALE) scheme without the need of any re-meshing. Furthermore, local mesh refinements around the interface are achieved using grid deformation methods, in which the number of elements is not increased. We show the motion of a freely moving particle suspended in a Giesekus fluid between two rotating cylinders. We investigate the effect of the Weissenberg number in this problem, and the effects of the mobility parameter and particle size on the migration of particles.

4:54PM EK.00004 The Structure of Nanoparticle Nucleation in Three-dimensional Wakes , NATHAN MURFIELD, SEAN GARRICK, University of Minnesota — Ultrafine, or nano-scale particles play an integral role in a wide variety of physical/chemical phenomena and processes and have application in microelectronics, chemical gas sensors, and atmospheric processes, to name a few. Accurate prediction of particle production rates and size distributions are of paramount importance in such processes. Physical measurement of particle nucleation and growth is difficult to observe in-situ and computation has the ability to shed light on the underlying physico-chemical dynamics. Direct numerical simulation of the homogeneous nucleation of dibutyl-pthalate (DBP) in turbulent, three-dimensional wakes is performed. The flows consistent of a "hot" mixture of DBP and nitrogen issuing into "cool" nitrogen. The effects of both large-scale and molecular mixing as well fluid turbulence on nano-sized nucleation are investigated under different flows. Additionally, the structure of nano-particle nucleation as well as the effect of the cooling rate on the size distribution of nucleating particles is investigated.

5:07PM EK.00005 Particle Focusing and Dispersion in Suspension Flow through a Corrugated Tube , GREGORY HEWITT, JEFFREY MARSHALL, University of Vermont — A computational study is performed of the transport of a particulate suspension through a corrugated tube using a discrete-element method. The tube is axisymmetric with a radius that varies sinusoidally, which in the presence of a mean suspension flow leads to periodic inward and outward acceleration of the advected particles. The oscillations in radial acceleration and straining rate lead to a net radial drift, with mean acceleration measuring an order of magnitude smaller than the instantaneous radial acceleration, which over time focuses small particles within the tube. The foundations of particle focusing in this flow are examined analytically using lubrication theory together with a low Stokes-number approximation for the particle drift. Computations are then performed using a finite-volume method for fluid flow in the tube at higher Reynolds numbers over a range of amplitudes, wavelengths and Reynolds numbers, examining the effect of each of these variables on the averaged radial fluid acceleration. A discrete-element method (DEM) is used to simulate particle behavior at finite Stokes numbers.

5:20PM EK.00006 Particle focusing in Lagrangian Coherent Structures , GUSTAAF JACOBS, San Diego State University — We discuss the relation between finite sized, inertial particle dispersion and Lagrangian Coherent Structures identified with recent Lagrangian visualization techniques. We consider the flow over a rectangular cylinder that features a vortex-dominated wake. Fluid particle transport barriers that attract fluid particles are visualized through contours of the Finite Time Lyapunov Exponent (FTLE) that measures the stretching of the fluid. Combinations of the transport barriers form Lagrangian Coherent Structures that are typically visualized through dye in experiments. Inertial particles closely follow these attracting transport barriers, as visualized by maxima in the FTLE field determined in backward time, for Stokes numbers smaller than unity. With increasing Stokes number the particles increasingly align with the transport barriers. At unity Stokes number when particles are well-known to exhibit particle focusing, large parts of a sharp particle streak are exactly aligned with the transport barrier in the FTLE field. The attracting transport barriers in the wake are hence excellent predictors of the location where particles focus. The inertial particles are shown to depart from the fluid particle transport barrier when criteria based on the stress tensor are met.

5:33PM EK.00007 Axial and lateral particle ordering in finite Reynolds number channel flows , PANDURANG KULKARNI, Levich Institute and Department of Chemical Engineering, The City College of New York, KATHERINE HUMPHRY, Department of Physics, Harvard University, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University, JEFFREY MORRIS, Levich Institute and Department of Chemical Engineering, The City College of New York — In pressure-driven channel flows at finite Reynolds number, suspended particles are known to undergo cross-streamline migration to a specific position off the centerline. We investigate the combined effects of channel geometry and particle concentration on the migration and ordering of particles in the channels with rectangular cross-sections using both experiments and numerical simulations. First we show that under confinement a single particle always migrates to the center of the longer dimension of the cross section for a range of cross-sectional aspect ratios. In case of a dilute suspension, the particles exhibit lateral and axial ordering owing to the hydrodynamic interactions of neighboring particles. Upon increasing the particle loading, a stepwise transition from one to two to many trains of particles in the lateral direction is observed. For a given channel cross-section, we present a criterion for this transition based on the number of particles per unit length.

5:46PM EK.00008 Hollow Vortices in Protoplanetary Disks Dynamically Stabilized by Trapped Dust Grains , XYLAR ASAY-DAVIS, Los Alamos National Lab, UC Berkeley, KATHERINE DECK, PHILIP MARCUS, UC Berkeley — We present 2D simulations of particle-laden vortices that dynamically maintain their hollow vorticity distribution (i.e., the vorticity is not a maximum at the vortex center). Vortices of this type may be the birthplaces of planets within in protoplanetary disks around newly formed stars. The vortices are embedded in a rotating, shearing Keplerian flow. Horizontally, the combination of the vortex flow and the Keplerian background flow drag dust grains, the building blocks of planets, to the center of the vortex. Vertically, grains may settle into the midplane of the disk, where the local gravity is zero, or they may be held aloft against gravity by an updraft within the vortex, just as hailstones are lofted in a thunderstorm. Our numerical simulations show that, as dust grains accumulate in the center of a vortex, the drag from the grains extracts angular momentum from the fluid flow, hollowing out the vortex. In the absence of dust grains, a hollow vorticity distribution is not stable; the vortex will readjust itself (sometimes violently) so that its vorticity decreases monotonically from the center. When dust is present, the vortex remains hollow in a dynamic equilibrium.

5:59PM EK.00009 Measurements of the Diameter and Velocity Distributions of Atomized Tablet-Coating Solutions for Pharmaceutical Applications[1] , KATHRYN OSTERDAY, UC San Diego, ALBERTO ALISEDA, U of Washington, JUAN LASHERAS, UC San Diego — The atomization of colloidal suspensions is of particular interest to the manufacturing of tablets and pills used as drug delivery systems by the pharmaceutical industry. At various stages in the manufacturing process, the tablets are coated with a spray of droplets produced by co-axial atomizers. The mechanisms of droplet size and spray formation in these types of atomizers are dominated by Kelvin-Helmholtz and Raleigh-Taylor instabilities for both low[1] and high[2] Ohnesorge numbers. We present detailed phase Doppler measurements of the Sauter Mean Diameter of the droplets produced by co-axial spray atomizers using water-based colloidal suspensions with solid concentrations ranging from fifteen to twenty percent and acetone-based colloidal suspensions with solid concentrations ranging from five to ten percent. Our results compare favorably with predictions by Aliseda's model. This suggests that the final size distribution is mainly determined by the instabilities caused by the sudden acceleration of the liquid interface. [1]Varga, C. M., et al. (2003) J. Fluid Mech. 497:405-434 [2]Aliseda, A. et al. (2008). J. Int. J. Multiphase Flow, 34(2), 161-175.

[1]Funding for this research project was provided by Pfizer, Inc.

6:12PM EK.00010 An evaporation model of colloidal suspension droplets[1] , SILVANA SARTORI, Mechanical and Aerospace Engineering Department, UCSD (California), AMABLE LIÑÁN, Escuela Técnica Superior de Ingenieros Aeronáuticos, UPM (Madrid, Spain), JUAN C. LASHERAS, Mechanical and Aerospace Engineering Department, UCSD (California) — Colloidal suspensions of polymers in water or other solvents are widely used in the pharmaceutical industry to coat tablets with different agents. These allow controlling the rate at which the drug is delivered, taste or physical appearance. The coating is performed by simultaneously spraying and drying the tablets with the colloidal suspension at moderately high temperatures. The spreading of the coating on the pills surface depends on the droplet Webber and Reynolds numbers, angle of impact, but more importantly on the rheological properties of the drop. We present a model for the evaporation of a colloidal suspension droplet in a hot air environment with temperatures substantially lower than the boiling temperature of the carrier fluid. As the liquid vaporizes from the surface, a compacting front advances into the droplet faster than the liquid surface regresses, forming a shell of a porous medium where the particles reach their maximum packing density. While the surface regresses, the evaporation rate is determined by both the rate at which heat is transported to the droplet surface and the rate at which liquid vapor is diffused away from it. This regime continues until the compacting front reaches the center of the droplet, at which point the evaporation rate is drastically reduced.

[1]Funded by Pfizer Inc.

Sunday, November 22, 2009 4:15PM - 5:59PM –
Session EL CFD III: Immersed Boundary Methods 200A

4:15PM EL.00001 Adaptive finite volume incompressible Navier–Stokes solver for 3D flows with complex immersed boundaries , GANESH NATARAJAN, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, University of Minnesota — We propose a generalisation of the CURVIB methodology (Ge & Sotiropoulos, JCP 2007) for the solution of the unsteady incompressible Navier–Stokes equations on arbitrary polygonal meshes in domains containing arbitrarily complex, moving immersed bodies. The new finite volume flow solver employs the hybrid staggered/non–staggered approach of Ge & Sotiropoulos (2007) in conjunction with generalised and robust discretisation procedures, so that it can be readily extended to handle adaptive meshes. The flow solver is combined with an isotropic adaptation strategy that effectively tracks flow features of interest and selectively enhances grid resolution. The resulting generic adaptive finite volume flow solver allows for computationally efficient, high resolution numerical simulations of a wide range of engineering and biological flows at Reynolds numbers much higher than what was possible with the original CURVIB methodology.

4:28PM EL.00002 Coherent-Structure Resolving Simulations of Turbulence in Natural Streams with the Curvilinear Immersed-Boundary Method[1] , SEOKKOO KANG, IMAN BORAZJANI, FOTIS SOTIROPOULOS, University of Minnesota — Critical prerequisite for developing a science-based approach to restoring natural streams is being able to model turbulence in real-life aquatic environments. We develop a powerful computational model for carrying out coherent structure resolving simulations of turbulent flows in natural streams at field scale conditions. The model employs the curvilinear immersed boundary method (CURVIB) of Ge and Sotiropoulos (J. Comp. Phys., 2007) to handle the arbitrarily complex channel geometries. To enable efficient simulations in the large-aspect ratio domains and highly stretched grids, arising due to the very small depth-to-length ratio of natural streams, we employ a fully implicit matrix-free GMRES method for the momentum equations coupled with a parallel algebraic multigrid method for the pressure Poisson equation. The capabilities of the method are demonstrated by carrying out high resolution LES as well as coarser resolution RANS simulations for the field scale meandering stream flow in the St. Anthony Falls Laboratory Outdoor StreamLab.

[1]This work was supported by NSF(Grant EAR-0120914 as part of the National Center for Earth-surface Dynamics and EAR-0738726) and the Minnesota Supercomputing Institute.

4:41PM EL.00003 An overset curvilinear/immersed boundary framework for high resolution simulations of wind and hydrokinetic turbine flows[1] , IMAN BORAZJANI, SURESH BEHARA, GANESH NATARAJAN, FOTIS SOTIROPOULOS, St. Anthony Falls Lab, University of Minnesota — We generalize the curvilinear/immersed boundary method to incorporate overset grids to enable the simulation of more complicated geometries and increase grid resolution locally near complex immersed boundary. The new method has been applied to carry out high resolution simulations of wind and hydrokinetic turbine rotors. An interior fine mesh contains the rotor blades and is embedded within a coarser background mesh. The rotor blades can be treated either as immersed boundaries or using curvilinear, boundary-conforming overset grids. The numerical methodology has been generalized to include both inertial and non-inertial frame formulations. The method is validated by applying it to simulate the flow for the NREL wind turbine rotor for various turbine operating points. Inviscid, unsteady RANS and LES simulations are carried out and compared with experimental data. Preliminary results will also be presented for the hydrokinetic turbine rotor installed at the Roosevelt Island Tidal Energy project in New York City.

[1]This research is supported by XCEL Energy RDF Grant RD3-42 and DOE's Advanced Water Power Projects program grant DE-FG36-08GO18168 and the Minnesota Supercomputing Institute.

4:54PM EL.00004 A second-order immersed boundary method with near-wall physics , RANDALL MCDERMOTT, NIST, CLARA CRUZ[1], University of Puerto Rico, KEVIN MCGRATTAN, NIST — The NIST Fire Dynamics Simulator is a variable-density large-eddy simulation code tuned for low-speed fire dynamics and heat transfer. The gas-phase numerics utilize a Cartesian staggered-grid arrangement and are generally second- order accurate. Until now, complex geometry has been treated with an immersed boundary method in which objects in the flow are forced to conform to the Cartesian grid. This method exactly corresponds to first interpolation method described in Fadlun et al. (J. Comp. Phys. 2000) and has been shown to exhibit zeroth order errors for non-conformal geometries. In the work presented here, we implement a variant on the second-order interpolation scheme of Fadlun et al. and extend the method to include the near-wall stress model of Werner and Wengle (8th Symp. Turb. Shear Flows 1991). To accomplish this we transform the first off-wall velocity into a streamwise coordinate system and update the components based on a momentum equation with a Werner and Wengle wall stress. The streamwise system components are then transformed back to the grid system to form the force terms required in the immersed boundary method. The new approach is tested in laminar and turbulent channel flow with the channel rotated through a range of angles relative to the Cartesian grid.

[1]NIST SURF student

5:07PM EL.00005 An immersed boundary method using the discrete streamfunction approach with an implicit forcing[1] , XING ZHANG, LNM, Institute of Mechanics, Chinese Academy of Sciences — Recently, Colonius and Taira (Comput. Methods Appl. Mech. Engrg. 2008) present an immersed boundary method in which the discrete streamfunction approach is used to solve the incompressible Navier-Stokes equations. In this talk, we proposed a novel immersed boundary method in the framework of the discrete streamfunction approach. Similar to the method of Colonius and Taira, the forcing term to impose the non-slip boundary condition is determined implicitly (by solving a linear system). However, in the present method, the linear system is in a simpler form and much easier to solve. The time to compute the forcing term is found to be negligible comparing with that to solve the Navier-Stokes equations. Some verifications and validations of this new method are presented. An accuracy test is first performed by using the decaying-vortex problem, both with and without the presence of immersed boundaries. Some canonical cases are then simulated, such as the flow over stationary and oscillating cylinders and flow over a vertical plate of zero thickness. All numerical results are in good agreement with those in literature. This is an indication that the method can handle boundaries of different shapes with acceptable accuracy.

[1]This work was supported by NSFC under the project no.10877201.

5:20PM EL.00006 A hybrid formulation to suppress the numerical oscillations caused by immersed moving boundaries , HAOXIANG LUO, HU DAI, PAULO FERREIRA DE SOUSA, Vanderbilt University — A family of immersed-boundary methods, based on the sharp-interface representation of the boundary and local interpolation/extrapolation, has been recently developed to handle complex and moving boundary problems encountered in biological flows. Implemented typically on structured meshes, these methods save the computational cost of grid generation and take advantage of efficient computations on structured grids. However, since some of the nodes near the immersed boundary do not have the regular finite-difference stencil available for discretizing the Navier-Stokes equation, a local interpolation or extrapolation scheme is often used to reconstruct the flow field around the nodes. The drawback of this approach is that when a non-stationary boundary moves across the mesh points, the change of the stencil for the solution reconstruction causes artificial oscillations in the pressure. To suppress the oscillations, we have introduced a set of hybrid nodes on which both the Navier-Stokes solution and flow reconstruction are sought, and they are weighted according to the distance to the immersed boundary. The method has been implemented in both two- and three-dimensional solvers to handle a class of biological locomotion problems including flow-structure interaction. The accuracy and capability of the solvers will be demonstrated.

5:33PM EL.00007 Immersed boundary method modeling of elastic capsules in superpositions of plane and axisymmetric extensional flows , ALEX SZATMARY, CHARLES EGGLETON, UMBC — Physics of elastic capsules in fluid is important to understanding biology, especially microcirculation, and is also useful in design of microfluidic devices. Linear flows, such as shear, plane extensional, and axisymmetric extensional flows, appear in many microfluidic applications, so understanding their effect on the deformation of capsules is necessary. The rotational component of the flow has no effect on deformation, and is neglected. The velocity gradient, not magnitude, leads to deformation, so the velocity magnitude is likewise neglected. Under eigendecomposition, any linear flow can then be represented as a superposition of a plane extensional and a plane axisymmetric flows, for analysis of capsule deformation. Computational modeling results using the immersed boundary method are reported.

5:46PM EL.00008 An Efficient Algorithm for Computing Physiological Fluid Flows , ANNE STAPLES, OMER SAN, Virginia Polytechnic Institute and State University — A new, efficient algorithm is presented for solving the one-dimensional model equations for internal physiological fluid flows. The models equations are a set of coupled, 1D nonlinear partial differential equations that govern the evolution of the pressure, velocity, and cross sectional area in internal biological fluid flows. The solution algorithm for these equations, the pulsed flow algorithm (PFA), begins with a partial asymptotic solution of the discretized model equations, then combines the equations and linearizes them, reducing them to tridiagonal form. The algorithm is applied to blood flow computations in the human arterial system and is compared to the most commonly used method for solving the model equations, the Lax-Wendroff (LW) scheme. The PFA algorithm is found to be approximately 40 times faster than LW for a single arterial segment, and approximately 6 times faster than LW for a model of the human arterial tree that includes the 55 largest arteries.

Sunday, November 22, 2009 4:15PM - 6:25PM –

Session EM Reacting Flows II 200B

4:15PM EM.00001 Laboratory-scale experiments to determine explosive properties using spherical concentric composite explosives , MATTHEW BISS, GARY SETTLES, Penn State University — Laboratory-scale air-blast experiments using gram-range composite explosive charges are presented. Composite charges consist of a spherical booster charge surrounded by a concentric spherical "candidate material" charge in the form of a shell. Air-blast explosive tests are conducted to measure the radius vs. time of the explosively-driven shock wave using digital high-speed shadowgraphy. Profiles of peak shock wave pressure vs. radius are then found using the Rankine-Hugoniot relationship for both the booster alone and the composite charges. Using calculated peak shock wave pressures, a procedure is developed to remove the booster effects from the signature produced by the composite charge, yielding the peak shock wave pressure effect due to the candidate explosive material alone. By this means we demonstrate the ability to properly characterize, at the laboratory scale with a few grams of explosive, insensitive explosive materials that require a booster charge for detonation. This characterization yields TNT equivalence and other useful explosive properties.

4:28PM EM.00002 Laboratory-scale blast testing of materials using air-shock loading , FORREST SVINGALA, MATTHEW BISS, MICHAEL HARGATHER, GARY SETTLES — The deformation of materials due to near-field explosions is a complex topic with many military and anti-terrorism applications. Typical explosive blast testing is conducted on large ballistic ranges using several kilograms of explosives, at a high cost, and with limited instrumentation. We have developed a range of experimental techniques for performing explosive blast research in the laboratory using gram-range explosive charges. Here we present recent measurements of the deformation of aluminum panels subjected to a range of explosive blast impulses. Through an explosive characterization procedure the blast impulse produced by a charge is known a priori. Our approach then couples the known explosion energy to the aluminum panels via shock wave propagation through the air. Time-resolved three-dimensional motion of the panel surface response is directly measured via stereoscopic high-speed digital image correlation. The data are used to compare material deformation characteristics to explosive loading parameters. These laboratory techniques can be used for validation of computational simulations and to develop high-strain rate material strength models. Ultimately results from these gram-range tests can be scaled to predict full-scale blast response characteristics.

4:41PM EM.00003 Planar Laser Induced Fluorescence of Shock Initiated Combustion of a Spherical Density Inhomogeneity , NICHOLAS HAEHN, CHRIS WEBER, JASON OAKLEY, MARK ANDERSON, DAVE ROTHAMER, RICCARDO BONAZZA, University of Wisconsin - Madison — A spherical density inhomogeneity with a stoichiometric mixture of H_2, O_2, and a diluent such as Xe is ignited with a planar shock wave. When a heavy bubble, such as Xe, is shock accelerated in a lighter ambient gas, such as Ar, the shock wave at the exterior periphery of the bubble travels faster than the interior transmitted wave, resulting in shock-focusing at the downstream pole of the bubble. The shock wave convergence results in a temperature much higher than the one behind the transmitted shock and auto ignition may occur at this location. For non-point source ignition experiments, the temperature is raised by a second shock acceleration from the planar shock that reflects from the shock tube's end-wall. These experiments shed light on the combustion characteristics under both turbulent and non-turbulent conditions. In addition, results are used for validating hydrodynamic codes with chemical reactions. The experiments are performed at the Wisconsin Shock Tube Laboratory in a 6 m vertical shock tube with a 25.4×25.4 cm^2 square cross-section. Diagnostics are performed using planar laser induced fluorescence of the OH^- molecule present during the combustion process. A Nd:Yag pumped dye laser at a wavelength of 283 nm excites the (1,0) band of the OH^- molecule.

4:54PM EM.00004 Modeling Unit Cell interactions for the Microstructure of a Heterogeneous Explosive: Detonation Diffraction Past an Inert Sphere[1] , D. SCOTT STEWART, JOHN B. BDZIL, University of Illinois, Urbana, IL — We describe an approach to model multi-phase blast explosive, which is primarily condensed y volume with inert embedded particles. The asymptotic theory of detonation shock dynamics governs the detonation shock propagation in the explosive. The detonation shock moves at a normal speed that depends on the shock curvature. The shock angle with the particle boundary is also prescribed. We describe theory to predict the behavior of a collection of such detonation shock/particle interactions in the larger aggregate. A unit cell problem, of a detonation shock diffracting over a sphere, is analyzed by analytical and numerical means. The properties of an ensemble of such unit cell problems are discussed with implications for the macroscopic limiting behavior of the heterogeneous explosive.

[1]Supported by AFRL Munitions Directorate, AFOSR Physical Mathematics.

5:07PM EM.00005 Axisymmetric Shock-Attached Frame Detonation Simulations , BRIAN TAYLOR, University of Illinois at Urbana-Champaign, ASLAN KASIMOV, Massachusetts Institute of Technology, D. SCOTT STEWART, University of Illinois at Urbana-Champaign — We present a method of simulating one-dimensional axisymmetric detonations governed by the reactive Euler equations in a reference frame moving with the shock surface. We use this methodology to verify relations between normal detonation speed, D_n, and curvature, (κ), as predicted by Detonation Shock Dynamics (DSD), an asymptotic theory derived in the limit of slowly varying, weakly curved detonations. Our simulations demonstrate the previously theorized instability of certain regions of D_n-κ solutions with multiple turning points, which result in rapid transition to strong detonation or failure of the reaction front. It is also shown that the shock-attached frame method can be used to obtain D_n-κ relations that satisfy the complete reactive Euler equations without restrictions to small curvature or near Chapman-Jouguet detonation speeds.

5:20PM EM.00006 Explosive, Spatially Distributed, Time Resolved Thermal Energy Deposition into a Finite Gas Volume , DAVID R. KASSOY, University of Colorado — Add thermal energy quickly to a finite volume of gas. Kaboom!! How much? How fast? An asymptotics-based analysis of the Navier-Stokes equations is used to study the response of a finite gas volume (length scale R') to spatially distributed, time resolved energy deposition per unit mass (characterized by q_R') on a specific time scale (t_H') short compared to the initial acoustic time of the volume (t_A'=R'/a_0'), where a_0'=$\sqrt{\gamma}$R'T_0 is the initial acoustic speed) such that t_H'/ t_A'=ε <<1. The initial state is described by (T_0',p_0',ρ_0'), and speed zero. The energy deposition is related to the initial internal energy by q_R'= [a_0']$^2/\alpha$ where $\alpha \leq$1, compatible with a characteristic temperature rise in the heated spot ΔT'/T_0'=O($1/\alpha$). Ephemeral inertial confinement prevails when ε^2 << α <<1, characterized by pressure rising with temperature because the density change and internal Mach number are both very small. Alternatively, when the energy addition reaches a critically large amount, $\alpha = \varepsilon^2$, the heat addition process is fully compressible and the internal Mach number reaches sonic values. One can apply these scaling concepts to explain the spontaneous appearance of hot spots observed in detonation initiation processes and the associated gasdynamic wave generation

5:33PM EM.00007 A two-phase micromorphic model for compressible granular materials , SAMUEL PAOLUCCI, WEIMING LI, JOSEPH POWERS, University of Notre Dame — We introduce a new two-phase continuum model for compressible granular material based on micromorphic theory and treat it as a two-phase mixture with inner structure. By taking an appropriate number of moments of the local micro scale balance equations, the average phase balance equations result from a systematic averaging procedure. In addition to equations for mass, momentum and energy, the balance equations also include evolution equations for microinertia and microspin tensors. The latter equations combine to yield a general form of a compaction equation when the material is assumed to be isotropic. When non-linear and inertial effects are neglected, the generalized compaction equation reduces to that originally proposed by Bear and Nunziato. We use the generalized compaction equation to numerically model a mixture of granular high explosive and interstitial gas. One-dimensional shock tube and piston-driven solutions are presented and compared with experimental results and other known solutions.

5:46PM EM.00008 Combined Space and Time Adaptivity Using the WAMR/G-Scheme Method , ZACHARY ZIKOSKI, SAMUEL PAOLUCCI, University of Notre Dame, MAURO VALORANI, University of Rome "La Sapienza" — Adaptive methods in CFD allow for savings in execution time by reducing the number of unknowns solved for at each computational step. The Wavelet Adaptive Multiresolution Representation (WAMR) provides spatial adaptivity which automatically supplies grid resolution based on the local demands of the solution itself. Likewise, the G-Scheme framework applies a similar capability in time. A system of ODEs can be ordered and separated into subsets of "fast" or near-equilibrium modes, "slow" or frozen modes, and "active" intermediate modes. Only the active modes are integrated, with asymptotics accounting for the contributions of the slow and fast dynamics. The pairing of spatial adaptivity using the WAMR method and temporal adaptivity using the G-Scheme allows for a substantial reduction in space-time degrees of freedom needed. The combined WAMR/G-scheme method is applied to several problems including reactive, compressible flow simulations.

5:59PM EM.00009 Simulations of Reacting Flow using Spectral Deferred Corrections , CANDACE GILET, ANN ALMGREN, JOHN BELL, MARCUS DAY, MIKE LIJEWSKI, Lawrence Berkeley National Lab, MICHAEL MINION, University of North Carolina at Chapel Hill — Numerical simulations of reacting flows frequently require capturing advection, diffusion, and reaction processes, which can have time scales that differ widely. When a fully explicit method is used the time step is controlled by the fastest process. This can result in calculations requiring so many time steps that the computational cost becomes prohibitively large. Fully implicit methods allow a larger time step, but require the simultaneous solution of (typically nonlinear) equations, again leading to restrictively high computational costs. Operator slitting methods allow for the use of a mix of implicit and explicit methods; however the splitting errors can be so large that prohibitively small time steps are still needed. An alternative to operator splitting is a class of methods called Spectral Deferred Corrections (SDC). The idea behind SDC methods is to represent temporal evolution of the system as an integral in time and develop algorithms that iteratively couple the different physical processes, thus reducing the splitting error. This work explores the use of SDC methodology in two-dimensional simulations of reacting flows with realistic chemistry. The results from simulations using SDC are presented and their performance is compared with that of Strang splitting.

6:12PM EM.00010 Modeling non-unity Lewis number effects in premixed flames , GUILLAUME BLANQUART, California Institute of Technology, ED KNUDSEN, HEINZ PITSCH, Stanford University — The combustion of hydrogen in Low Swirl Burners (LSB) is considered as an alternative for power production for it is characterized by low emissions and high efficiency. However, lean hydrogen premixed flames are subject to thermo-diffusive instabilities induced by the large diffusivity of hydrogen. The numerical modeling of these flows remain challenging for the transition of small scale instabilities into large scale turbulent structures cannot be modeled by conventional theories. In this work, a model is presented for the simulation of premixed flames with non-unity Lewis number fuels. This model relies on the Levelset/Progress variable approach which was found perfectly suited for the modeling of premixed flames with close to unity Lewis number fuels such as methane. Combined with the solution of an additional transport equation for mixture fraction, this model is formulated and validated in simple 1D laminar premixed flames. The model is found to capture accurately global quantities such as burning velocity and flame thickness as well as mixture fraction fluctuations. Then, this model is applied in Large Eddy Simulation (LES) of a Low Swirl Burner of H_2/Air ($\phi = 0.4$). The simulation shows the formation of a strongly wrinkled flame with local extinction. The results obtained with this new formulation show significant improvement when compared with experimental measurements.

Sunday, November 22, 2009 4:15PM - 6:25PM –

Session EN Experimental Techniques III: Scalar 200C

4:15PM EN.00001 Lifetime Based Temperature and Velocity Measurement using High-Speed Camera and Temperature Sensitive Particles , SATOSHI SOMEYA, MITSUNORI UCHIDA, KAORU TOMINAGA, KEIKO ISHII, YANRONG LI, KOJI OKAMOTO, Univ. of Tokyo — A simultaneous measurement technique of velocity and temperature distribution in a fluid flow was developed. Particle image velocimetry (PIV) was combined with a lifetime based scalar measurement technique. A measurement method based on the luminescent lifetime is appropriate for detecting the transient temperature field. Developed method needs a high speed camera, a laser (one excitation wave length) and temperature sensitive particles (TSParticles). The luminescence from TSParticles doped with EuTTA was detected by the high speed camera, which was not equipped with any image intensifier, at 15000 frames per second. Imaging at the high frame rate has a possibility to carry out PIV with a wide dynamic range. A decay of luminescent intensity was detected in detail. The decay curves at various temperature conditions were fitted well to exponential functions, from which the decay constant at each temperature was obtained. The proposed technique was applied to measure the temperature and the velocity field in a natural convection driven by Marangoni force and by buoyancy in a rectangular tank. The accuracy of the temperature measurement of the proposed technique was $\pm 0.5^{\circ}$C.

4:28PM EN.00002 Magnetic Resonance Imaging Measurements of Film Cooling Effectiveness , CHRISTOPHER ELKINS, MARCUS ALLEY, JOHN EATON, Stanford University — Film cooling through holes and slots is used extensively in gas turbine engines to protect combustor walls, stator vanes, and turbine blades. Film cooling effectiveness has been shown to depend on myriad geometrical and flow parameters. Magnetic Resonance Velocimetry (MRV) and Concentration (MRC) measurements efficiently acquire entire 3D velocity and scalar information making them well suited to investigate the large design parameter space. In addition, MRV and MRC provide information in the film supply plenum and hole which is traditionally unobtainable. Here, MRC is extended to near wall measurements to determine film cooling effectiveness. Measurements are made for a single film hole (d=5.8 mm, l/d=4 and α=30°) in the wall of a square channel. Velocity and scalar concentration data are presented for multiple blowing ratios. The data show the evolution of vortices around the jets, the coolant mixing, and the 2D film cooling effectiveness distribution. Measurements within the plenum and injection hole show the origin of specific flow structures.

4:41PM EN.00003 Validation of LES predictions with Microscopic Particle Image Velocimetry in an axial symmetric Confined Impinging Jets Reactor , EMMANUELA GAVI, Iowa State University, IA, MATTEO ICARDI, DANIELE MARCHISIO, Politecnico di Torino, Italy, MICHAEL G. OLSEN, RODNEY O. FOX, Iowa State University, IA, DJAMEL LAKEHAL, ASCOMP GmbH — Micromixer devices, such as the Confined Impinging Jets Reactor (CIJR) are under study, in particular for micro- and nanoparticles precipitation processes. In this work the flow field in an axial symmetric CIJR was studied by means of microscopic Particle Image Velocimetry (microPIV), an innovative experimental technique that allows to measure the instantaneous flow field over a global domain. Experimental measurements were carried out at four flow rates, ranging from quasi-steady laminar to unsteady turbulent regime. To the authors knowledge this work is the first one to report experimental microPIV data on an axial symmetric microscopic device. Measurements were then employed to validate predictions obtained with Large Eddy Simulation (LES). CFD results were able to reproduce the instability of the flow observed in the CIJR and a good quantitative agreement is found for both the mean velocity components and the fluctuations. An interesting result of this study is the understanding of the necessity of mimic the pumping instability in LES simulations of the CIJR, by imposing appropriate inflow boundary conditions, which contribute to determine the unsteady behavior of the flow.

4:54PM EN.00004 Visualization of turbulent reacting flow in a microscale nanoprecipitation reactor , YANXIANG SHI, SOMASHEKAR VISHWANAT, MICHAEL OLSEN, RODNEY FOX — A flow visualization technique using the pH sensitive dye phenolphthalein was used to visualize and quantify turbulent reacting mixing in a microscale nanoprecipitation reactor. Phenolphthalein is colorless at pH lower than 8, but turns pink at higher pH, making it useful for visualizing acid-base reactions. Using this dye, turbulent reactive mixing in a confined impinging jets reactor (CIJR) was investigated. The reactor has two inlet streams, one at a pH of 3, and the other at a pH of 11. Phenolphthalein is also dissolved in both streams. A flash lamp with a extremely short pulse duration is used to freeze the turbulent motion of the fluids, and images are captured using a video camera. Quantitative mixing data are obtained by using a thresholding technique where local image intensities are transformed to binary signals which represent the local pH: 0 stands for pH lower than 8 and 1 for pH higher than 8. For each Reynolds number under consideration, thousands of realizations are acquired. Using this thresholding technique, probability density functions are obtained, allowing comparison to numerical simulations.

5:07PM EN.00005 Determination of Plasma Electron Density from Optical Measurements , BRIAN NEISWANDER, ERIC MATLIS, THOMAS CORKE, University of Notre Dame — Plasma has been shown to be effective in many flow control applications, but now may also find use in adaptive optics. Plasma's index of refraction is coupled with it's electron density which may be adjusted for adaptive control. An experimental setup to verify the relation between plasma electron density, pressure, and voltage is presented. A non-thermal DBD plasma cell is created by evacuating air and applying a voltage potential between two conducting glass slides. Plasma forms in the chamber between the glass and the applied voltage potential controls the electron density. A HeNe laser is passed through the plasma cell and then focused onto a duo-lateral position sensing device (PSD). The plasma cell is oriented at an angle to the laser's beam and so changes in the plasma's index of refraction produce lateral translations in the beam position. Differences in the PSD output with and without plasma provides for the calculation of the electron density averaged over the beam spot area. The data from this experiment will be used to further develop an adaptive plasma lens for wavefront aberration corrections.

5:20PM EN.00006 Photochemistry of tracers used for molecular tagging diagnostics[1] , R. NEHE, R. BASU, A.M. NAGUIB, M.M. KOOCHESFAHANI, Michigan State University — Phosphorescence characteristics of molecular tracers are important to certain class of molecular tagging diagnostics, including the recently demonstrated technique of molecular tagging manometry (MTM). The photoluminescence of tracers acetone and biacetyl are studied in an effort to understand the factors affecting their phosphorescence lifetime. The phosphorescence emission of these tracers is known to be strongly quenched by oxygen. Here we present a more complete picture of quenching by accounting for the effects of self-quenching, in addition to oxygen quenching. It is found that acetone self quenching explains the factor of order ten reduction in lifetime observed in our oxygen-free experiments. Biacetyl phosphorescence, on the other hand, is not significantly affected by self quenching. The order of magnitude reduction in biacetyl lifetime measured in our oxygen-free experiments is connected to triplet-triplet annihilation reaction occurring at high intensities of the excitation light.

[1]This work was supported by National Science Foundation Grants No. CTS-0649744 and CHE-0714028

5:33PM EN.00007 Design Feasibility Study of Whole-Field Pressure Measurements in Gas Flows: Molecular Tagging Manometry (MTM)[1] , M.M. KOOCHESFAHANI, R. BASU, A.M. NAGUIB, Michigan State University — We present the theoretical foundation, implementation framework and experimental demonstration of a new diagnostic technique for non-intrusive, whole-field measurement of pressure within gasses. The new technique, which is referred to as Molecular Tagging Manometry (MTM), relies on oxygen quenching of phosphorescence emission from photo-excited tracers in oxygen-containing gases. As the pressure increases, the density of oxygen becomes larger, leading to a shorter emission lifetime: a working principle that is similar to pressure sensitive paint (PSP) but applied within the body of the flow. Using an experimental apparatus that is built around a pressure vessel, the viability of MTM is demonstrated for the first time using acetone as a tracer.

[1]This work was supported by NSF SGER Grant # CTS0649744.

5:46PM EN.00008 Liquid crystal thermometry for micro-fluidic applications[1] , TAIT POTTEBAUM, University of Southern California — Liquid crystal thermometry has been implemented in a micro-channel and the performance of the technique quantified. Implementation of the technique is subject to constraints on imaging and illumination configurations similar to the constraints on micro-PIV. In addition, the proximity of the measurements to interfaces and surfaces from which light scatters leads to high noise levels that cannot be reduced by wavelength filtering (such as with fluorescent particles) because the temperature information is contained in the color of the particles. Therefore, circular polarization filtering is used, exploiting the circular dichroism of the thermochromic liquid crystal (TLC). Encapsulated TLC particles were flowed through the micro-channel and subjected to a series of uniform temperatures for calibration. To validate the technique, a temperature gradient was imposed with no flow. Finally, the technique was applied to micro-channel flow with an imposed wall temperature gradient in the flow direction. Liquid crystal thermometry can now be applied to a wide range of micro-fluidic applications.

[1]This work is supported by NSF Grant CBET-0748294.

5:59PM EN.00009 Effect of Experimental Parameters and Image Noise on the Error Levels in Molecular Tagging Velocimetry/Thermometry (MTV/T) , JIANGHUA KE, Clarkson Unversity, DOUGLAS BOHL, Clarkson University — In this work the effect of experimental parameters on the error levels associated with simultaneous measurement of velocity and temperature using Molecular Tagging Velocimetry/Thermometry (MTV/T) are quantified via simulated images. Images were simulated using Gaussian profile laser lines. Noise was added to the images using a uniform random distribution and a Gaussian random distribution to simulate electronic noise and shot noise respectively. The results showed that the error levels in the velocity and temperature measurements were inversely related for most experimental parameters including the laser line thickness, fluid temperature and image delay times. It is concluded that the dynamic range of the technique depends on the flow speeds and temperatures and must be determined for each experiment individually. Error levels, for 95% confidence, were found to be better than 0.3° C for temperature and 0.2 pixels for velocity given typical experimental parameters.

6:12PM EN.00010 Measuring near-wall temperatures using dual-tracer fluorescence thermometry and evanescent-wave illumination[1] , MYEONGSUB KIM, MINAMI YODA, Georgia Institute of Technology — Fluorescence thermometry measures liquid temperatures based on changes in fluorescence intensity. Dual-tracer (or ratiometric) fluorescence thermometry (DFT) improves the accuracy of FT by taking the ratio of the emissions from two different fluorescent species excited at the same wavelength by the same illumination, thereby removing changes in fluorescence intensity due to spatial variations in the excitation. Moreover, DFT using two species with opposite temperature sensitivities can significantly increase the sensitivity of the technique. The ratio of the signals from an aqueous solution of fluorescein (FI) and sulforhodamine B (SrB), which have intensities that increase and decrease, respectively, when volumetrically illuminated at 514 nm, varies by as much as 7% per $^\circ$C for fluid temperatures T = 15-60 $^\circ$C. The method has experimental uncertainties, based on temperature calibrations obtained with volume illumination, of $\pm$1.1 $^\circ$C and $\pm$0.3 $^\circ$C at spatial resolutions of 3.7 μm and 30 μm, respectively. This talk describes extending DFT to near-wall temperature measurements by using evanescent-wave illumination to measure fluid temperatures averaged over about the first 0.5 μm next to the wall, or in most cases, the wall surface temperature.

[1]Supported by NSF and ONR.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session EP Instability: Jets & Wakes II 200D

4:15PM EP.00001 Near Wake Flow Topology of a Blunt Trailing Edge Profiled Flat Plate , LAKSHMANA SAMPAT DODDIPATLA, ARASH NAGHIB LAHOUTI, PhD Candidate, The University of Western Ontario, HORIA HANGAN, Professor, The University of Western Ontario, KAMRAN SIDDIQUI, Associate Professor, The University of Western Ontario — Wake flows behind two dimensional bodies are unstable due to formation of spanwise von Karman vortices accompanied by three dimensional streamwise instabilities, also referred to as rolls and ribs, respectively. These three dimensional instabilities lead to two distinct instability modes (Mode A and Mode B), or a combination of the two, depending on the flow Reynolds number and the profile geometry. It has been observed that the ribs wrap around the rolls, progressively distorting them. Therefore, enhancing the action of streamwise vortices can lead to early suppression of the spanwise von Karman vortices accompanied by the reduction of fluctuating lift and base drag. The present investigation seeks to better understand these near wake instabilities for blunt trailing edge profiled bodies of various aspect ratios, for flow Reynolds numbers ranging from Re(d)=500 to Re(d)=2200, and various inlet conditions. Planar Laser Induced Fluorescence (PLIF) visualizations and measurements are performed in the near wake to study and characterize the topology of streamwise and spanwise vortices.

4:28PM EP.00002 ABSTRACT WITHDRAWN –

4:41PM EP.00003 Bifurcations in axisymmetric wakes and their stabilization by base bleed[1] , ENRIQUE SANMIGUEL-ROJAS, PATRICIO BOHÓQUEZ, ALEJANDRO SEVILLA, CARLOS MARTÍNEZ-BAZÁN, Universidad de Jaen — We investigate the instability properties of the laminar incompressible flow around a cylindrical body with a rounded nose and length-to-diameter ratio 2, at zero angle of attack, combining experiments, three-dimensional direct numerical simulations and a global linear stability analysis. The direct numerical simulations and the global stability results are in excellent agreement in their prediction of a first stationary and three-dimensional bifurcation in the wake, which takes place at $Re_{c1} \approx 325$. Moreover, both the experiments and the numerical simulations show the existence of a second oscillatory bifurcation at $Re_{c2} \approx 400$. However, although the global stability analysis does also predict the existence of an oscillatory bifurcation, it is at a considerably larger value of $Re \approx 518$. The disagreement between the global linear analysis and the experimental and numerical results in the prediction of the oscillatory bifurcation is investigated and justified in terms of the flow field used to performed the stability analysis. In addition, we report the existence of two critical values for the bleed coefficient, defined as the bleed-to-free-stream velocity ratio, to re-stabilize the wake to its axisymmetric steady state.

[1]Supported by the projects P07-TEP-02693 and DPI2008-06624-C03-02.

4:54PM EP.00004 Surface signatures of momentum/zero-momentum wakes in stratified fluids[1] , S.I. VOROPAYEV, H.J.S. FERNANDO, C. NATH, Arizona State University — The momentum (over- and under-thrusted) wakes of self-propelled bodies in stable density stratified fluids were studied using scaled laboratory experiments and theoretical analysis. Particle image velocimetry (PIV) and a highly sensitive cooled Infra-Red (IR) camera were used for flow diagnostics. Two problems of broad interest that have not been received adequate study were the focus of this research: (i) thermal (IR) surface signatures of wakes of submerged/surface self-propelled bodies; and (ii) thermal (IR) and hydrodynamic (PIV) surface signatures of submerged/surface turbulent jets. The measurements and modeling delineated the mechanisms of vertical turbulent transport, the types of flow structures formed at the surface and their dependence on bulk wake properties on governing parameters. The physics-based models and parameterizations so developed help extrapolate laboratory results to oceanic environments.

[1]This research was supported by the ONR.

5:07PM EP.00005 Vortex statistics in a bluff body wake with varying inlet conditions[1] , BENGT E.G. FALLENIUS, JENS H.M. FRANSSON, Linne Flow Centre, KTH — For many years the flow in the wake behind bluff bodies have been the subject of extensive studies. One reason is the interesting flow phenomena that occur therein, such as the von Kármán vortex shedding. Another is the reduced pressure that contributes highly to the drag force. Learning how to control the vortices and increasing the pressure in the wake can lead to reduction of noise and vibrations in structures as well as improved energy efficiency for vehicles. This study investigates experimentally how changes of the boundary layer on the bluff body, i.e. the inlet condition for the wake flow, affects the vortex structures in the wake. The bluff body boundary layer is modulated by applying varying strengths of continuous suction or blowing through the surface of the bluff body, which is made of a permeable material. A high number of instantaneous velocity fields of the wake flow have been acquired by means of particle image velocimetry. Each velocity field has then been processed by a computer program, developed in order to detect, analyze and store information about the characteristics of small-scale vortices. Statistics for the vortex structures in the bluff body wake have then been compared for different inlet conditions.

[1]This work was financially supported by the Swedish Research Council and the Göran Gustafsson Foundation.

5:20PM EP.00006 Physical reduced model for the flow past a circular cylinder for $47 < \boldsymbol{Re} < 100$[1] , IAGO C. BARBEIRO, IVAN KORKISCHKO, JULIO R. MENEGHINI, J.A.P. ARANHA, NDF, Escola Politécnica, University of São Paulo, Brazil — The *Reynolds* ($Re = \frac{U.D}{\nu}$) range considered for this study lies within the time-periodic bidimensional régime where any experiment started by its stationary solution should evolve to a stable limit-cycle. This transient oscilatiory ramp starts with the exponential growth of the linear unstable eigenmode and finishes bounded by nonlinear effects with multi-harmonics extra dissipation. The steady solution and the leading eigenmode are numerically obtained using FEM discretization (*Taylor-Hood P2/P1 elements*) and *Arnoldi* iterations, then the nonlinear evolution operator is employed to generate new modes complementing the linear eigenmode up to a given order. The full NSE is then projected onto this physical base (*nonlinear Galerkin projection*) leading to a physical reduced system. This reduced model has a simple framework to track many nonlinear features like meanflow evolution and energy changes between the harmonics, clarifying the nonlinear mechanisms that takes this system to a periodic orbit. Numerical and experimental (*Particle Image Velocimetry*) evidences will be presented at the time of the meeting.

[1]Authors have grants from FINEP-CTPetro, FAPESP and Petrobras.

5:33PM EP.00007 Floquet stability analysis of the wake of a circular cylinder undergoing VIV , RAFAEL S. GIORIA, JULIO R. MENEGHINI, NDF, Dept. Mech. Eng., Escola Politécnica, University of São Paulo — In some flow situations, bluff bodies undergo vortex-induced vibration (VIV). The amplitude of oscillation of a circular cylinder undergoing vortex-induced vibration has a three branch response in respect to the reduced velocity ($V_R = U_\infty/df$, where U_∞ is the free stream velocity, d the cylinder diameter and f the natural frequency of the oscillation of the cylinder). One remarkable feature of the behavior is the jump in the amplitude between the upper and the lower branch: hysteresis is observed on this jump. This hysteretic phenomena is still subject of investigation. In this work, we investigate the circular cylinder three-dimensional wake behavior on the three different branches of response with the intention of acquiring better understanding of the hysteretic jump on the amplitude response of VIV. In order to realize this investigation, Floquet stability analysis of the two-dimensional wake of a circular cylinder in VIV is conducted. We choose two Reynolds numbers (Re) close to the secondary transition observed on the wake of a circular cylinder undergoing imposed oscillations: 280 and 300. The stability analysis is realized for each of the three branches of VIV response. We expect to observed subcritical unstable modes in the hysteretic region as these modes also present hysteresis on the transition.

5:46PM EP.00008 Characterizing cylinder and hydrofoil wake dynamics , MORTEN KJELDSEN, Norwegian University of Science and Technology (NTNU), BJARTE G. SEIM, NTNU, ROGER E.A. ARNDT, University of Minnesota — A number of high speed PIV measurements of wakes trailing a NACA 0015, c=0.081m, and a cylinder D= 0.0127m, in the speed range 2 through 9 m/s have been made in the high speed water tunnel at SAFL- UMN. The cylinder vortex shedding follows closely St=0.2, while that off the hydrofoil is more irregular. Although the hydrofoil shows a more irregular nature a measure for both shedding frequencies and vortex strength of is of great interest also for drag analysis. The direct approach mapping individual structures, e.g. vorticity based analysis, can be obscured by the quality of the measurements made, hence other methods to reveal frequency and strength are in demand. A study were the mapping of time variation of the main flow direction impulse flux integrated over the wake at, minimum two, downstream positions has been made. A cross-correlation analysis of the impulse flux can reveal structure transport speeds, the frequency spectrum will reflect the shedding frequency, while the temporal variation represents the strength. For the hydrofoil it's shown that a significant cross- correlation is present. In terms of spectra even the more structured shedding from cylinders are hard to capture, and finally the vortex strength found using the cited algorithm seems somewhat unreliable. A thorough comparison between the suggested measure and traditional measures is given.

5:59PM EP.00009 A New Mechanism for Mixing Enhancement in Turbulent Mixing Layer[1] , WEI ZHAO, GUIREN WANG, University of South Carolina — It is well known that in shear layer, even under the most effective modes—sub-harmonic modes, the mixing under active forcing is only nearly two times enhanced compared to that without forcing, and saturation will be reached for the further increased forcing amplitude. Hence, the potential of the mixing enhancement is limited for active forcing based on receptivity and instability in mixing layer. Recently we observed that, in a confined mixing layer in a pipe, such saturation could be overcome, so that extremely fast mixing enhancement could be achieved by further increase of the forcing amplitude. However the mechanism behind the high receptivity and fast mixing is still not clear. Preliminary data shows that the optimized forcing frequency does not change with Reynolds number based on bulk flow velocity and pipe diameter in the range of moderate Reynolds number. Therefore we postulate that the phenomenon might be related to the resonance frequency of the pipe flow and the frequency corresponding to the fast mixing augmentation should be scaled with some Strouhal number. The dependence of the optimized frequency on various parameters is investigated.

[1]Supported by the NSF RII funding (EPS-0447660).

6:12PM EP.00010 Reduced Order Modeling for Beam Propagation through a Shear Layer , JURGEN SEIDEL, CASEY FAGLEY, STEFAN SIEGEL, THOMAS MCLAUGHLIN, US Air Force Academy — The performance of airborne platforms emitting or receiving light beams is severely hampered by the flow field around the turret mounted on the air vehicle. From a fluid dynamics point of view, the flow separating from the turret develops large, coherent structures. The goal of this research is to improve system performance by mitigating these structures using feedback flow control. While developing a feedback flow control system is a multi-step process, the most important step is the design of a Reduced Order Model of the flow field under consideration. A blowing and suction slot is used to actuate the flow field. Three dimensional simulations have been resolved and show a large amount of controllability of the optical path difference (OPD). Proper orthogonal decomposition techniques are then applied to open loop simulation data. A low dimensional model is realized by non linear system identification techniques. The accurate model of the flow field is then utilized to develop control strategies to mitigate the optically detrimental coherent structures and simulate closed loop behavior of the flow field. The chosen control algorithm will be simulated in a CFD environment for verification.

Sunday, November 22, 2009 4:15PM - 6:25PM –

Session EQ Instability: Interfacial and Thin-Film III 200E

4:15PM EQ.00001 Study of Pattern Evolution in Nanoscale Viscous Films Subject to Pillar Instability by Thermocapillary Stresses , SANDRA TROIAN, MATHIAS DIETZEL, California Institute of Technology, 1200 E. California Blvd., MC 128-95, Pasadena, CA 91125 — Nanofilms of molten polymer subject to a large transverse thermal gradient are known to undergo instability to arrays of elongated pillars separated by a few microns. The simplicity with which such large area patterns can be formed offers a new type of resistless, non-contact lithography in which pillar spacing and amplitude can be controlled by plate separation distance, initial film thickness and thermal gradient. Here, we investigate the evolution of pillar formation from the linear to non-linear regime by examining an unexplored limit of Benard instability accessible to films of nanoscale dimensions. Given the small plate spacings used in experiment, we examine the influence of initial conditions on the wave number selection process. For intermediate values of plate separation, the pillar spacing is well predicted by linear stability analysis and unaffected by the type of initial disturbance applied. For larger plate separation, the pillar arrays evolve toward hexagonal symmetry while subsequent depletion of fluid in the interstitial regions leads to formation of subharmonic protrusions. These bifurcations are suggestive of cascade processes observed in other thin film evolution equations.

4:28PM EQ.00002 Influence of Temperature Dependent Viscosity on the Evolution of Pillar Instabilities in Polymer Nanofilms , MATHIAS DIETZEL, SANDRA TROIAN, California Institute of Technology, 1200 E. California Blvd., MC 128-95, Pasadena, CA 91125 — We have previously shown that evolution of periodic elongated pillar arrays in nanoscale polymer films subject to a significant transverse thermal gradient represents an extreme limit of Benard-instability. Such ultrathin films allow access to a regime in which destabilizing thermocapillary forces outweigh stabilization by capillary and gravitational forces by many orders of magnitude. In prior work, the melt viscosity was assumed constant. Here we explore the influence of increasing viscosity as the nanopillars evolve to approach a cooler target. A comparison of models incorporating either a linear or exponential variation of viscosity with temperature with previous results reveals that the pillar spacing obtained from linear stability analysis is only marginally affected. Full numerical simulations beyond the linear regime, however, indicate a more noticeable influence on pillar shape and even more so, pillar growth times. The influence of thermal viscous effects on this moving boundary problem is therefore critical to estimates of processing times for technological applications.

4:41PM EQ.00003 Effect of insoluble surfactant on the evolution of electrified falling films , B. UMA, University of Pennsylvania, R. USHA, Indian Institute of Technology Madras — Nonlinear stability of a thin liquid film down an inclined plane when the film surface is contaminated with an insoluble surfactant and the film is subjected to an electric field has been analyzed. The evolution equation of the electrified contaminated film has been derived using long-wave perturbations. Linear stability analysis of the electrified contaminated film reveals that the effect of electric Weber number and Marangoni number is to destabilize the film flow system. The nonlinear evolution of the film thickness is found by numerically solving the coupled non-linear evolution equation for the film thickness and the surfactant concentration in a periodic domain. Evolution profiles of the thin film thickness and the surfactant concentration down an inclined plane are plotted for the different values of parameter governing the flow.

4:54PM EQ.00004 Convective Structure and Heat Transfer of Liquid Films Evaporating into a Pure Vapor Environment[1] , J.T. KIMBALL, J.C. HERMANSON, University of Washington, J.S. ALLEN, Michigan Technological University — The stability, convective structure and heat transfer of upward facing, evaporating, thin liquid films were studied experimentally. Dichloromethane, n-pentane, and methanol films initially 5 mm to 50 μm thick were subjected to constant or impulsive superheat levels. The films resided on a temperature controlled, gold-plated copper plate in a closed, degassed test chamber. The dynamic film thickness was measured at multiple discrete points using ultrasound and instability wavelength and convective structure information was obtained by schlieren imaging. For films below the transition Rayleigh number there is little convective heat transfer present within the film. In films above this transition, the Nusselt number increases with increasing Rayleigh number. The transition in the heat flux occurs over a wide range of Marangoni numbers. Transient experiments reveal an initial rise in heat flux due to evaporation, followed by a decrease and then increase at the onset of convective motion.

[1]This research was supported by NSF grant CBET-0651755.

5:07PM EQ.00005 Rupture of the interface between two fluids in presence of surface agents , MATTHIEU ROCHE[1], MOUNIR AYTOUNA[2], DANIEL BONN, HAMID KELLAY — Rupture of interfaces between two fluids is ubiquitous in everyday life. It is well established that this process is driven by three stresses: capillarity, viscosity and inertia. The effects of viscosity and inertia have been widely studied, whereas capillary effects have attracted less attention. We present results concerning the rupture of an interface in the presence of surface agents. Ruptures in the presence of a surfactant (SDS) and in the presence of a mixture of surfactant (SDS) and polymers (PVA) have been studied. In both cases, surface agents are diluted in the outer fluid. All experiments have been carried out using a microfluidic flow-focusing device. In the former case, the thinning dynamics of the neck behind the droplet exhibits two modes of thinning, depending on the external flow rates, and the surfactant concentration. These modes can be related to dynamic surface tension phenomena. In the latter case, the thinning slows down exponentially in the late instants. Structures analogue to beads-on-a-string also develop. This is very similar to what is reported for the breakup of polymeric jets, though polymers are outside the jet in our case.

[1]CPMOH, Universite Bordeaux 1
[2]Van der Waals Zeeman Institute, Amsterdam

5:20PM EQ.00006 Drops and waves on a viscous film coating a vertical fiber , CAMILLE DUPRAT, FREDERIQUE GIORGIUTTI-DAUPHINE, Laboratoire FAST, Orsay — We consider the nonlinear dynamics of a liquid viscous film flowing down a vertical fiber. The initially flat film spontaneously breaks up into a regular wavetrain due to the Rayleigh-Plateau instability. The characteristics of the fully developed structures then depend on the dominant mechanisms. When the advection of the waves by the flow dominates over the instability, we observe moderate amplitude waves that propagate without carrying mass. When surface tension is strong, the film breaks up into large amplitude drops exhibiting a recirculation region. The transition between drops and waves regimes and the corresponding structures have been characterized experimentally and numerically. When both surface tension and viscosity effects are strong, the system can be seen has a paradigm for active (unstable), dissipative and dispersive media. The particular shape of the resulting structures gives rise to complex interactions (attraction and repulsion) leading to the formation of bound states. The system then appears to select a finite number of preferred specific distances in agreement with a weak interaction theory developed by D. Tseluiko, S. Saprykin and S. Kalliadasis (Imperial College London).

5:33PM EQ.00007 Tear Film Dynamics for Blink Cycles with a Wetting Cornea and Evaporation , R.J. BRAUN, J. TANG, L.P. COOK, University of Delaware, D.M. ANDERSON, George Mason University — We consider a lubrication model in one spatial dimension for the human tear film. The ends of the domain move to mimic the blink cycle of the eyelids; the underside of the film is wettable surface due to a conjoining pressure and evaporation occurs from the top surface of the film. The film can equilibrate at a small finite thickness where the conjoining pressure and evaporation balance. This small thickness is interpreted as a dry spot; the model can compute past dry spot formation. Results are given for the film dynamics with either pressure of flux boundary conditions on the ends of the film. The film can form dry spots for some boundary conditions of either type. The persistence of the dry spots with respect lid motion and other effects are explored.

5:46PM EQ.00008 Turbulent gas – laminar liquid flows , DMITRI TSELUIKO, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, UK — We study two-dimensional co-current and counter-current turbulent gas-laminar liquid flows. Specifically, we consider a liquid film flowing under gravity down the lower wall of an inclined channel when a turbulent gas flows above it. The solution of the full system of equations describing the gas-liquid flow faces serious technical difficulties. However, a number of certain assumptions allow separating the gas problem and solving it independently. This permits finding perturbations to pressure and tangential stresses at the interface. We then proceed to the liquid problem and derive model equations describing the dynamics of the interface, i.e. boundary-layer equations, an integral-boundary-layer model and a long-wave model and we analyze solutions of these equations. As the simplest prototype retaining all the important physical mechanisms, we derive a weakly nonlinear model, a Kuramoto-Sivashinsky-type equation with a dispersive term and a contribution from the turbulent gas.

5:59PM EQ.00009 Spin-coating of layered liquids , ALAN MCINTYRE, LUCIEN BRUSH, University of Washington — An axisymmetric model of the spin-coating of two immiscible, vertically stratified liquids is derived using lubrication theory. The model includes gravitational, van der Waals, capillary, viscous forces and evaporation/condensation processes. During the evolution of uniform layers, the lower layer thins monotonically and never reaches zero thickness. With evaporation the upper layer disappears in finite time. With condensation the upper layer reaches a steady-state thickness. Fully nonlinear calculations, including viscous and evaporation/condensation effects, show that disturbances of the lower layer have a greater effect on the upper layer than disturbances of the upper layer have on the lower layer. Disturbances along the upper gas-liquid free surface propagate outward more rapidly than those along the lower liquid-liquid interface. The effects of additional forces on the evolution of bilayer films during spin-coating are also presented.

6:12PM EQ.00010 Linear and nonlinear stability of floating viscous sheets , GILLES PFINGSTAG, IJLRDA UPMC Paris 6 / LPS Ecole Normale Supérieure, AREZKI BOUDAOUD, LPS Ecole Normale Supérieure, BASILE AUDOLY, IJLRDA UPMC Paris 6 — The dynamics of thin viscous sheets is relevant to industrial processes such as float glass and to natural processes such as plate tectonics. We study the behavior of a thin, Newtonian viscous sheet undergoing stretching and bending. We use asymtotic expansions to derive the equations governing the evolution of the thickness and of profile of the sheet subjected to an external force field. Two models are obtained according to the scaling of the characteristic evolution time. In this framework, we investigate the stability of a viscous sheet floating on a denser fluid [at rest], accounting for gravitation and surface tension.The various instable modes are described. A nonlinear analysis yields the long-time evolution of the sheet. We also discuss possible extensions to falling sheets or to variable viscosities.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session ER Convection I 200F

4:15PM ER.00001 Search for the "ultimate state" in turbulent Rayleigh-Bénard convection[1] , GUENTER AHLERS, Dept. of Physics, UCSB, DENIS FUNFSCHILLING, LSGC CNRS, Nancy, EBERHARD BODENSCHATZ, MPI for Dyn. and Selforg., Göttingen — Measurements of the Nusselt number Nu will be reported for turbulent Rayleigh-Bénard convection of a cylindrical sample. They cover the Rayleigh-number range $10^{11} \lesssim Ra \lesssim 2 \times 10^{15}$ using N_2 ($Pr = 0.72$) and SF_6 ($Pr = 0.79$ to 0.84) at pressures up to 19 bars and near-ambient temperatures. The sample had a height $L = 2.24$m and diameter $D = 1.12$m and utilized the high-pressure vessel known as the "Uboot of Göttingen" at the Max Planck Institute for Dynamics and Self-Organization in Göttingen, Germany. For $Ra \lesssim 4 \times 10^{13}$ the data yielded $Nu \propto Ra^{\gamma_{eff}}$ with $\gamma_{eff} = 0.308$ and did not show the transition near $Ra = 10^{11}$ to an "ultimate regime" that was reported by Chavanne et al. At $Ra = 4 \times 10^{13}$ there is a well defined but continuous transition to a regime where γ_{eff} is smaller than 0.30.

[1]Supported in part by NSF Grant DMR07-02111, the Max Planck Society, and the Volkswagen Stiftung.

4:28PM ER.00002 Mean temperature profiles in turbulent Rayleigh-Benard convection , O. SHISHKINA, M. KACZOROWSKI, DLR Goettingen, A. THESS, TU Ilmenau, C. WAGNER, DLR Goettingen, Germany — We report a study of mean vertical temperature profiles (TPs) in turbulent Rayleigh-Benard convection of water, $Pr = 4.38$, in unit-aspect-ratio cylindrical and cubic cells for Ra up to 10^9, based on DNS. The Nusselt numbers Nu computed for cylindrical cells are found to be in excellent agreement with the experimental data by Funfschilling et al. [J. Fluid Mech., vol. 536 (2005), pp. 145-154]. Based on this validation, the DNS data are used to extract TPs. In the DNS for the cylindrical geometry, reported in Shishkina & Thess [J. Fluid Mech. (2009), in press], we find that near the heating and cooling plates the TP $\Theta(y)$ obey neither a logarithmic nor a power law. We show that the Prandtl–Blasius BL theory predicts the TP-shapes with an error 7.9% within the thermal BLs alone. We further show that the profiles can be approximated by a stretched exponential approximation (SEA) of the form $\Theta(y) \approx 1 - \exp(-y - 0.5y^2)$ with an absolute error $< 1.1\%$ within the thermal BLs and $< 5.5\%$ in the whole cell. We provide also more accurate analytical approximations of the profiles involving higher-order polynomials in the SEA. Further, based on the DNS in a cube we extract TPs and estimate the accuracy of the above SEAs. Finally, we construct and analyze the quality of SEAs, which are based on local Nu.

4:41PM ER.00003 Scaling laws in turbulent Rayleigh-Benard convection with different geometry , HAO SONG, PENGER TONG, Hong Kong University of Science and Technology — The discovery of scaling laws in the heat flux, large-scale circulation and temperature statistics in turbulent convection has stimulated considerable experimental and theoretical efforts, aimed at understanding the universal nature of the observed scaling laws. Because of historical reasons, most of the experimental results were obtained in upright cylindrical cells with small aspect ratios. An important question one might ask is: To what extend are these scaling laws universal in that they are independent of the cell geometry? Understanding of this question has important implications to large-scale astro/geophysical convection, such as that in the atmosphere and oceans, in which boundary effects are less important. In this talk, we report an experimental study of turbulent convection in a horizontal cylinder with the bottom 1/3 (curved) surface heated and the top 1/3 surface cooled. The experiment is carried out with varying aspect ratios and Rayleigh numbers. It is found that the measured Nusselt number and Reynolds number obey the same scaling laws as those obtained in the upright cylinder. The local temperature statistics, on the other hand, change with the aspect ratio of the cell and are different from the earlier results. The experiment reveals important geometric effects of the scaling laws in turbulent convection. *Work supported by the Research Grants Council of Hong Kong SAR.

4:54PM ER.00004 Shallow moist Rayleigh-Bénard convection with piecewise linear equation of state[1] , JOERG SCHUMACHER, THOMAS WEIDAUER, TU Ilmenau, OLIVIER PAULUIS, Courant Institute, New York University — An idealized framework to study the impacts of phase transitions on atmospheric dynamics is presented. Condensation of water vapor releases a significant amount of latent heat, which directly affects the atmospheric temperature and density. Here, phase transitions are treated by assuming that air parcels are in local thermodynamic equilibrium, which implies that condensed water can only be present when the air parcel is saturated. This reduces the number of variables necessary to describe the thermodynamic state of moist air to three. It also introduces a discontinuity in the partial derivatives of the equation of state. A simplified version of the equation of state is obtained by a separate linearization for saturated and unsaturated parcels. When this equation of state is implemented in a Boussinesq system, the buoyancy can be expressed as a piecewise linear function of two buoyancy variables, D and M, and height z. Numerical experiments in this setting allow then to study transitions from cumulus to stratocumulus clouds.

[1]The work is supported by the Deutsche Forschungsgemeinschaft and the US National Science Foundation.

5:07PM ER.00005 Extensive Scaling of Computational Homology and Karhunen-Loève Decomposition in Rayleigh-Bénard Convection Experiments , HUSEYIN KURTULDU, MICHAEL SCHATZ, Georgia Institute of Technology — We apply two different pattern characterization techniques to large data sets of spatiotemporally chaotic flows in Rayleigh-Bénard convection (RBC) experiments. Both Computational homology (CH) and a modified Karhunen-Loève decomposition (KLD) are used to analyze the data. The KLD dimension D_{KLD}, the number of eigenmodes required to capture a given fraction of the eigenvalue spectrum, is computed for different subsystem sizes. A similar quantity D_{CH} for the same experimental data is acquired by the probability distribution of topological states constructed from the outputs of CH. We show that both D_{CH} and D_{KLD} scale over a large range of subsystem sizes for the state of SDC; moreover, we find the presence of boundaries leads to deviations from extensive scaling that are similar for both methodologies.

5:20PM ER.00006 Deterministic flow reversal in thermal convection , DETLEF LOHSE, Twente, KAZU SUGIYAMA, Tokyo, RUI NI, Hongkong, ENRICO CALZAVARINI, Lyon, SIEGFRIED GROSSMANN, Marburg, TAK SHING CHAN, Twente, HENG DONG XI, SHENG-QI ZHOU, Hongkong, CHAO SUN, Twente, KEQING XIA, Hongkong — Two-dimensional (2D) numerical simulations of the Boussinesq equations are presented with Rayleigh numbers up to $Ra = 10^9$ and aspect ratio of about 1. They reveal a diagonal large scale convection roll ("wind of turbulence") and smaller rolls in the two remaining corners diagonally opposing each other. These corner flow rolls play a crucial role for the mechanism of large scale wind reversal: They grow in kinetic energy and thus also in size thanks to the plume detachments from the boundary layers up to the time that they take over, leading to the breakdown of the large scale convection roll and the formation of a new one, rotating in the other direction. Based on the numerical simulations and on theoretical arguments we identify the characteristic time scale for this whole process. According to present precision it seems to grow with Ra as a power law. – Employing PIV techniques the same deterministic flow reversal mechanism is shown to be at work for thermal convection in quasi 2D rectangular Rayleigh-Benard cells.

5:33PM ER.00007 Benard convection in the presence of micro particles , LAYACHI HADJI, University of Alabama — We study Benard convection in water containing a small volume fraction of micro particles. The investigation is motivated by recent experiments of natural convection of aqueous suspensions [1] conducted at an average temperature of 20 degrees C in which the authors report a decrease in Nusselt number compared to pure water. This effect has been attributed to density inversion in the base state taking place near the lower boundary caused by the sedimentation of the aluminum oxide particles, the density of which is greater than that of water. We attempt to elucidate these findings by carrying a stability analysis on a model of convection for a liquid suspension having a nonlinear equation of state. The model accounts for the coupled effects of Brownian motion, sedimentation and thermophoresis. The balance of the latter yields a nonlinear base profile for the concentration of particles. Density inversion occurs near either the lower or the top boundary depending on the balance between sedimentation and thermophoresis and on the size and density of the particles. Parameter range for the onset and stability of the resulting double layer convection is given and the implications the results may have on the heat transfer in nanofluids are discussed.

[1] B. H. Chang, A.F. Mills, E. Hernandez, Int. J. Heat Mass Transfer, 51 (2008) 1332-1341.

5:46PM ER.00008 Refined measurements on the structure of thermal boundary layers in turbulent Rayleigh-Benard convection[1] , ANDRE THESS, RONALD DU PUITS, CHRISTIAN RESAGK, Ilmenau University of Technology — We present highly resolved temperature measurements at a large- scale Rayleigh-Benard experiment simultaneously undertaken at the top and the bottom plate using small microthermistors. For the first time the temperature measurements have been complemented by local heat flux measurements at the surface of both horizontal plates using special heat flux sensors. The experimental facility used for this purpose is an adiabatic cylinder with an inner diameter of D=7.15 m filled with air. An electrical heating plate at the bottom and a free hanging cooling plate at the top trigger the convective flow. The work reported here is limited to the H=6.30 m where the shape of the global flow is well known. We will discuss results of measurements of profiles of the mean temperature and compare the behaviour of the top and bottom boundary layers.

[1]Supported by the Deutsche Forschungsgemeinschaft

5:59PM ER.00009 Pattern Control and State Estimation in Rayleigh-Bénard Convection[1] , ADAM PERKINS, MICHAEL SCHATZ, Georgia Institute of Technology — We report on a new experimental approach to study instability in Rayleigh-Bénard convection. The convective fluid absorbs incident infrared laser light, thereby altering the fluid flow. Rapid scanning of the light allows nearly simultaneous actuation at many spatial locations of the pattern. This approach is used to impose reproducibly a given convection pattern. Control is demonstrated by preparing repeatedly a pattern near a straight roll instability. Selected perturbations are applied to this ensemble and decay lifetimes are measured as the system relaxes to the base state. We find that decay lifetimes increase near the instability and give a quantitative measure of distance from instability. We also create patterns that undergo the instability, giving a set of systems evolving from nearby initial conditions on both sides of the instability boundary. This set can be used to test systematically the sensitivity of state estimation, a crucial process in forecasting. Preliminary results of applying one state estimation algorithm to these diverging pattern trajectories will be discussed.

[1]This work is supported by the National Science Foundation.

6:12PM ER.00010 Turbulent Rayleigh-Bénard Convection with Conductive Plates , RALF WITTENBERG, Simon Fraser University — Despite considerable experimental, theoretical and numerical effort, for turbulent Rayleigh-Bénard convection in the high-Ra limit the scaling of the enhanced bulk heat transport, measured by the Nusselt number Nu, with the temperature drop across the fluid, given by the Rayleigh number Ra, is still incompletely understood. While most work has assumed a fixed temperature drop across the fluid, it has recently become clear that this assumption is mathematically and experimentally inadequate, and in the quest to reconcile theory and experiment the influence of the finite conductivity of the bounding plates has been receiving increasing attention. We review recent progress in this area, and discuss in particular rigorous variational bounds on the Nu-Ra scaling for finite Prandtl number convection for general thermal boundary conditions ranging between the fixed temperature and fixed flux extremes, including the case of plates of finite conductivity. We show in particular that the usual fixed temperature assumption is a singular limit of the full problem, while in the large-Ra limit, we find a bound of Nu $\leq R^{1/3}$, where R is a Rayleigh number in terms of the temperature drop across the full fluid-plate system.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session ES Geophysical: Oceanographic I 200G

4:15PM ES.00001 Fate and Transport of Fluid Mixed at the Boundary of a Lake , CHRIS REHMANN, DANIELLE WAIN, MICHAEL KOHN, JOSHUA SCANLON, Iowa State University — Several studies have shown that fluxes in stratified water bodies are controlled by turbulence and mixing at sloping boundaries, but fewer have investigated the fate of the mixed fluid. We conducted field experiments in two Iowa lakes to track fluid mixed on the sloping sides as it moved into the interior of the lake. Measurements from a meteorological station characterize the forcing, while measurements from thermistor chains and acoustic profilers provide information on the resulting internal waves. Tracer measurements combined with microstructure profiling illustrate the boundary mixing and transport of mixed fluid. In one case in the smaller of the two lakes, turbulence created by currents from long internal waves mixed fluid that propagated from the shore as an intrusion governed by a balance between inertia and buoyancy. In another case, the transport is more likely due to a jet corresponding to the second vertical mode of the internal waves. Recent results from a larger lake will be used to determine the effect of lake size and geometry on the transport.

4:28PM ES.00002 Direct Numerical Simulation of a Model Estuary , ROLF HENNIGER, LEONHARD KLEISER, ETH Zurich, ECKART MEIBURG, UC Santa Barbara — We investigate mixing and sedimentation processes in laboratory-scale estuaries by means of high-resolution Navier-Stokes simulations. A positively buoyant, sediment-laden freshwater river is considered that enters a notional ocean, upon which particles sediment out at the lower boundary of the current. The computational setup, while employing a simplified geometry, accounts for the most important features of a typical estuary. The flow is studied in a spatially developing framework that allows us to obtain statistically stationary solutions for freshwater/saltwater mixing rates and particulate settling profiles. Details of the particle settling process are investigated both during the initial transient phase, as well as for statistically stationary conditions. We observe qualitatively good agreement of the settling mechanisms with corresponding laboratory experiments. The properties of the particulate plume in the freshwater current are analyzed as a function of the particle Stokes settling velocity.

4:41PM ES.00003 The distribution of age in a coastal river plume , ALEXANDER HORNER-DEVINE, MELYSSA NAGAMINE, SHAUN BEVAN, YEPING YUAN, University of Washington — Fluid age and residence time are arguably the most important variables for predicting changes in water quality in aquatic systems; however, no existing experimental technique can directly resolve them, and our understanding of the relationship between the distribution of age and fluid mechanical principals is incomplete. We present a novel experimental technique for determining the spatial distribution of the age of fluid parcels in laboratory flows. We use the technique to investigate the retention of fluid in a coastal river plume using a series of experiments on a 2m diameter rotating table. In this system, an anticyclonic eddy, or bulge, sets up near the river mouth and accumulates approximately half of the water discharged from the river. Initially, fluid collects continually in the core of the bulge such that the maximum age in the eddy after 10 rotation periods is approximately 6 rotation periods. Subsequently, the bulge becomes baroclinically unstable and the aged core is distorted by smaller coherent eddies. The instability breaks down the retention process and transports fluid from the aged core into the coastal current. The instability also appears to periodically modulate the partitioning of new fluid between the bulge and the coastal current, resulting an coherent pulses of fluid in discreet age classes in the age distribution.

4:54PM ES.00004 Internal-Tide Scattering by 2D Topography: Experimental Study , MATTHIEU MERCIER, Ecole Normale Supérieure de Lyon, Laboratoire de Physique, THOMAS PEACOCK, Massachusetts Institute of Technology, Department of Mechanical Engineering, THIERRY DAUXOIS, Ecole Normale Supérieure de Lyon, Laboratoire de Physique — Scattering of internal tides is an important mechanism to understand mixing and energy transfer in the ocean. Numerical and oceanographic studies have shown that topographies can be responsible of conversion from low modes to higher modes, leading to transfer from large to smaller scale inducing local mixing and higher damping rate along propagation. To understand and quantify more precisely this mechanism, we generate a mode-1 internal tide using a new configuration for the wavemaker recently developed by Gostiaux $et\ al.$. Experiments have been realized at the Coriolis Turntable in Grenoble (France), a cylindrical rotating tank of 13 m diameter. The velocity fields observed using PIV technique are analyzed in terms of modal decomposition. Knowing that 97% of the generated internal tide energy flux is associated to a mode-1 internal tide, we analyze its interaction with a 2D gaussian topography. Subcritical and supercritical bathymetry are considered according to the frequency of the incoming internal tide. Estimations of the amount of reflected energy by the topography, such as the scattering into higher modes of the transmitted wave field, are in good agreement with numerical predictions.

5:07PM ES.00005 Rippled Bed Evolution over Wave Groups: Implications to Bottom Roughness Calculations , DIANE FOSTER, SYLVIA RODRIGUEZ ABUDO, University of New Hampshire — Outside of the surf zone, seabed ripples are the source of significant dissipation of free surface gravity energy. The dissipation is a function of both the ripple geometry and the hydrodynamic forcing. Observations of natural irregularly rippled beds and the oscillatory, two-dimensional, time-varying velocity field were collected using a submersible Particle Image Velocimetry (PIV) system in both a full-scale and smaller-scale environments. The full-scale observations were obtained over natural sand beds, whereas the smaller-scale observations were obtained over artificial low specific gravity beds. Bedform evolution regimes characterized by the ripple radius of curvature were examined relative to measures of the non-dimensional bed stress (the grain roughness Shields parameter), the non-dimensional pressure gradient (the Sleath parameter), and the water column coherent structure formation from the ripples (the swirling strength). Anorbital bedforms were found to respond to individual waves by modulating amplitude as wave groups passed. These observations suggest that the bottom dissipation due to movable sediment beds may be more dynamic than previously assumed.

5:20PM ES.00006 LES of an oscillating current over an inclined slope , BISHAKHDATTA GAYEN, SUTANU SARKAR, University of California, San Diego — Large-Eddy Simulations (LES) are performed to investigate the dynamics of a stratified bottom boundary layer on a continental slope under a tidal current. Turbulent mixing is observed to be different between the upslope and downslope phases of the flow. The observed difference is found to be related to the phase-dependent modulation of the stratification.Flow instabilities and turbulence in the bottom boundary layer excite internal gravity waves that propagate far from the wall region. The wave field during the upslope and downslope flow exhibits significant differences. The slope angle is varied and found to play an important role in determining properties of the bottom boundary layer.

5:33PM ES.00007 Settling of porous particles through a pycnocline , KOLJA KINDLER, MIT, ARZHANG KHALILI, MPI for Marine Microbiology, ROMAN STOCKER, MIT — Downward carbon flux in the Ocean is largely governed by particle settling. Most marine particles are highly porous and settle at low Reynolds numbers. We present results of an experimental investigation of porous spheres settling through a thin density interface at $O(0.1) < Re < O(1)$. We tracked 100–500μm hydrogel spheres with porosities exceeding 90% and small hyrodynamic permeabilities, for which the exchange of interstitial fluid is driven purely by diffusion. Two limiting scenarios are identified based on the particle's initial excess density $\Delta\rho$ relative to the lower (denser) fluid phase. For large positive $\Delta\rho$ we observe a drag enhancement similar to that reported for non-porous spheres. For small negative $\Delta\rho$ interstitial fluid exchange by diffusion dominates the dynamics at the pycnocline, leading to retention of particles at the pycnocline. These results could contribute to better estimate the vertical carbon flux and explain high particle concentrations frequently observed at pycnoclines.

5:46PM ES.00008 3D experimental investigation of vortex dipoles in shallow water , JULIE ALBAGNAC, Institut de Mecanique des Fluides de Toulouse, PIERRE BRANCHER, OLIVIER EIFF, LAURENT LACAZE, FREDERIC MOULIN, IMFT — Vortex dipoles are often associated to sediments transport. Shallow dipoles have often been expected to exhibit Quasi-2D dynamics. However, recent lab experiments have shown the existence of a horizontal spanwise vortex at the front of the dipole. It is of great importance to describe and quantify their dynamics. We investigate dipole evolution and spanwise vortex formation. Reproducible laminar dipoles are created with flap apparatus. For fixed flaps gap, shallowness of the dipole and dipole propagation velocity are the parameters controlling the flow. PIV measurements were performed in two horizontal planes and the vertical symmetry one for a wide range of control parameters. These measurements led us to define criteria on the spanwise vortex generation and dynamics. Complete 3D geometry and dynamics of the dipole are obtained using 3D-3C scanning PIV. Transition from a Q-2D dipole to a 3D dipole considering the spanwise vortex has been enhanced thanks to this powerful technique.

5:59PM ES.00009 Turbidity Currents in Complex Topographies , MOHAMAD NASR-AZADANI, SEBASTIAN SAEGELER, MICHAEL ZOELLNER, ECKART MEIBURG, UC Santa Barbara — We consider particle-laden gravity currents interacting with complex seafloor topographies, such as mini-basins, ridges or meandering channels. Both two- and three-dimensional Navier-Stokes simulations are employed in order to investigate their dynamics, entrainment and depositional behavior for a range of flow and geometrical parameters. We observe that coherent vortical structures generated by topographical effects can lead to the formation of strong nonuniformities in the sediment deposit. Results from a parametrical study are discussed, based on two-dimensional simulations of depositing currents produced by a lock-exchange configuration flowing through a minibasin, in order to quantify the effects of the geometrical parameters and particle settling speed on the sediment deposit fields. The over-spill and lateral deposit profile is studied for flows passing through meandering channels with the continuous inflow and outflow from the system.

6:12PM ES.00010 The inhibiting effect of stratified mixing on surface-stress-driven flow in a cylinder , C.P. CAULFIELD, BPI & DAMTP, University of Cambridge, A. SHRAVAT, AOPP, University of Oxford, C. CENEDESE, WHOI — We extend previous work of Boyer, Davies & Guo (1997: Fluid Dyn. Res. Vol. 21 pg 381-401) to consider the evolution of an initially two-layer stratified fluid in a cylindrical tank which is driven by a horizontal rotating disc. The flow induced by the disc drives entrainment at the interface, and the layer nearer to the disc deepens. Through high-frequency conductivity probe measurements, we establish that the deepening layer is well-mixed, and the interface depth between the two evolving layers appears to be largely constant. Under certain circumstances, we find that the rate of increase in depth of the deepening layer decreases with time, at variance with the results of Boyer et al, and implying that the characteristic velocities in the deepening layer decrease as the upper layer deepens. Such imperfect, decaying spin-up is a natural consequence of the total energy budget of the flow, as the combined power requirements of entrainment and layer homogenization can inhibit the characteristic velocities of the deepening layer approaching the (constant) velocities of the driving disc, as assumed by Boyer et al. We investigate the dependence of both the entrainment and the efficiency of the mixing on the external parameters of the flow, in particular the bulk Richardson number defined in terms of the initial layer depth, interfacial reduced gravity, and disc azimuthal velocity.

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session ET Vortex Dynamics and Vortex Flows III 200H

4:15PM ET.00001 Quantum turbulence[1] , D.P. LATHROP, M.S. PAOLETTI, M.E. FISHER, University of Maryland, K.R. SREENIVASAN, International Centre for Theoretical Physics — Long range quantum order underlies a number of related physical phenomena including superfluidity, superconductivity, the Higgs mechanism, Bose-Einstein condensates, and spin systems. While superfluidity in Helium-4 was one of the earliest discovered of these, it is not the best understood, owing to the strong interactions (making theoretical progress difficult) and the lack of local experimental probes. Approximately three years ago, our group discovered that micron-sized hydrogen particles may be used to label quantized vortices in flows of superfluid helium. Particles not on vortices trace the motion of the normal component of the superfluid. This ability has given a new perspective on an old subject. By directly observing and tracking these particles, we have directly confirmed the two-fluid model, observed vortex rings and reconnection, characterized thermal counterflows, and taken local observations of the very peculiar nature of quantum turbulence.

[1]Supported by grants NSF DMR-0606252 and NSF DMR-0906109.

4:28PM ET.00002 Velocity statistics in superfluid and classical turbulence , K.R. SREENIVASAN, International Centre for Theoretical Physics, D.A. DONZIS, University of Maryland and Texas A&M, M.E. FISHER, D.P. LATHROP, M.S. PAOLETTI, University of Maryland, P.K. YOUNG, Georgia Institute of Technology — Past work, summarized in part by Vinen & Niemela (*J. Low Temp. Phys.* **129**, 213 (2002)) and by Walmsley *et al. Phys. Rev. Lett.* **99**, 265302 (2007)), suggests that similarities exist between superfluid and classical turbulence. Conversely, the more recent work of Paoletti *et al.* (*Phys. Rev. Lett.* **101**, 154501 (2008)) has highlighted differences: in particular, the probability density function (PDF) of the turbulent superfluid velocity, measured by tracking the trajectories of small hydrogen particles, is strongly non-Gaussian with power-law tails, in contrast to classical homogeneous and isotropic turbulence for which the PDF is nearly Gaussian. Here, we explore this dichotomy. Since the observed power-law exponent of -3 in the superfluid case can be traced to the reconnection of quantized vortices, it is natural to explore the role of vortex reconnection in the classical case. We surmise that the latter, if it is significant at all, must involve vortices of high intensity. Using direct numerical solutions of homogeneous and isotropic turbulence on a grid of linear size 4096, we condition the velocity statistics on the magnitude of vorticity and find that the resulting conditional PDFs, if normalized on their own standard deviation, remain Gaussian for all vorticity magnitudes.

4:41PM ET.00003 Dynamics of the Lattice Array Formation in Superfluid Helium , KRISTINA GAFF, University of Maryland, College Park, ENRICO FONDA, Universitá di Trieste; University of Maryland at College Park, MATTHEW PAOLETTI, DANIEL LATHROP, University of Maryland, College Park, KATEPALLI SREENIVASAN, International Centre for Theoretical Physics; University of Maryland at College Park — In 1955, Feynman theorized that the lowest energy state of rotating superfluid helium would result in a lattice structure of quantized vortices. In 1979, Yarmchuk *et. al.* observed a small lattice array of up to eleven vortices using clusters of ions, and later experiments observed the lattice array in superfluid ^{4}He perpendicular to the axis of rotation using hydrogen ice particles (Bewley *et. al.* 2006). Yet, the formation and dynamics of these arrays remain otherwise unexplored experimentally. Now, by visualizing sub-micron particles trapped on quantized vortices, we characterize the lattice array formation in superfluid helium. Our study investigates the lattice formation by independently varying rotation rate and temperature.

4:54PM ET.00004 Visualization of quantized vortices near the λ-transition using nanoparticles[1] , ENRICO FONDA, Università di Trieste - University of Maryland, KRISTINA T. GAFF, MATTHEW S. PAOLETTI, University of Maryland, KATEPALLI R. SREENIVASAN, International Centre for Theoretical Physics - University of Maryland, DANIEL P. LATHROP, University of Maryland — The dynamics of an irregular tangle of quantized vortices in superfluids, i.e. quantum turbulence, has recently drawn interest for its connections with different areas of research and for the premise of shedding light on classical turbulence. Previous experiments (Bewley et al. 2006, Paoletti et al. 2008) studied the superfluid flows in ^{4}He using micron-sized solid hydrogen particles as tracers. Because of their size, Stokes drag does not allow them to stay trapped on quantized vortices close to the λ-transition, where the trapping potential is weaker. A new technique has been discovered to create and visualize sub-micron particles. Several size estimates of these nanoparticles have been made based on both optical and fluid dynamical properties. Being smaller, but not small enough to be influenced by thermal motions, the particles are more passive and are less affected by Stokes drag. Thus they stay trapped closer to transition and on faster moving vortices. Preliminary results from near-transition observations are presented.

[1]Work supported by NSF-DMR 0606252 and NSF-DMR 0906109.

5:07PM ET.00005 Elementary vortex processes in thermal superfluid turbulence , DEMOSTHENES KIVOTIDES, LOUISE WILKIN, University of California, Santa Barbara — By solving pertinent mathematical models with numerical and computational methods, we analyze the formation of superfluid vorticity structures in a turbulent normal fluid with an inertial range exhibiting Kolmogorov scaling. We demonstrate that mutual friction forcing causes quantum vortex instabilities whose signature is spiral vortical configurations. The spirals expand until they accidentally meet metastable, intense normal fluid vorticity tubes of similar curvature and vorticity orientation that trap them by driving them towards low mutual friction sites where superfluid bundles are formed. The bundle formation sites are located within the tube cores, but, due to tube curvature and many-tube interaction effects, are displaced by variable distances from the tube centerlines as they follow the contours of the latter. We analyze possible implications of these processes in fully developed thermal superfluid turbulence dynamics.

5:20PM ET.00006 Scaling and symmetry breaking in a vortex dipole , M. DURAN-MATUTE, F. FONTENELE ARAUJO, R.R. TRIELING, G.J.F. VAN HEIJST, Fluid Dynamics Laboratory, Dept. of Applied Physics, Eindhoven University of Technology — A vortex dipole is experimentally studied in a layer of salt water driven by time independent electromagnetic forcing. In particular, we characterize the flow along the dipole axis by measuring the Reynolds number Re as a function of the Chandrasekhar number Ch (the ratio of Lorentz to viscous forces). We find $Re \sim Ch^{\alpha}$, with the scaling exponent α ranging from $\alpha = 1$ (viscous regime) to $\alpha = 1/2$ (advective regime). The underlying transition emerges as a symmetry breaking of the axial flow, which we quantify via the skewness of the axial velocity profile.

5:33PM ET.00007 Three-dimensional stability of dipole-wall interactions , CHRISTOPHER SUBICH — The interaction of a dipole colliding with a no-slip boundary has been well studied in two dimensions at both low (Orlandi, 1990) and moderate (Kramer, 2007) Reynolds numbers, and just such a case is advocated as a benchmark for numerical simulations (Clercx, 2006). However, in three-dimensional flow the vortex pair has a short-wavelength elliptic instability (Leweke, 1997) that ultimately destroys the two-dimensional character of the vortex pair. For small enough perturbations, however, a dipole can complete a partial rebound from a no-slip wall before the elliptic instability dominates its evolution. The small-scale vorticity generated in the collision creates a different character to the instability. This work presents the results of three-dimensional pseudospectral numerical simulations of the instability at a Reynolds Number of 1,250.

5:46PM ET.00008 Computing vortex states with Dirac constraints[1] , P.J. MORRISON, The University of Texas at Austin, G.R. FLIERL, Massachusetts Institute of Technology — A procedure for calculating vortex states that uses Dirac constraint theory will be described. Several examples will be given, including V-states, rigidly rotating m-fold symmetric vortex patches. The technique is general and applies to both nonconvex and convex contours, and both barotropic and baroclinic vortex dynamics.

[1]PJM is supported by US Department of Energy Contract No. DE-FG03-96ER-54346.

5:59PM ET.00009 Vortex Dipoles with Prescribed Nonlinear Profiles , ALAN ELCRAT, KEN MILLER, TRENTON ALBRECHT, Wichita State University — We obtain translating vortex pairs using a generalization of the method used in Elcrat,Fornberg,Horn and Miller JFM2000 409 for vortex patches in flow past a cylinder. When the profile function F in $\Delta\psi = \omega F(\psi - \alpha)$, ψ the stream function for the flow, is piecewise constant the flows obtained are the Translating V-States found in Wu,Overman and Zabusky JCP1984 53. When the vortex support attaches to the symmetry axis the flows are called dipoles, and when F is linear we retrieve the Lamb-Chaplygin pair. The solution procedure that we use in which ψ is obtained from an iteration $\Delta\psi_{n+1} = F(\psi_n)$ where the area of the vortex support is fixed in an inner iteration, allows general F. We have computed solutions for F that have 2 continuous derivatives at $\psi = 0$. These lead to "smooth" dipoles, and the solutions obtained have elliptical shapes of the sort obtained recently in Kizner and Khvoles Reg.Chaotic Dyn. 2004 9. The methods that we use extend naturally to translating dipoles in the β -plane approximation for flow over a rotating sphere.

6:12PM ET.00010 Oscillatory boundary layer in a dipole vortex driven by a periodic electromagnetic force , SERGIO CUEVAS, ALDO FIGUEROA, EDUARDO RAMOS, Universidad Nacional Autonoma de Mexico — The laminar boundary layer flow driven by an oscillatory Lorentz force in a shallow electrolytic layer is analyzed experimentally and theoretically. The force is produced by the interaction of an injected alternate electric current and the magnetic field of a small dipole magnet, externally attached to the bottom wall of a plexiglass container. Alternate currents with frequencies and amplitudes in the range of 10-200 mHz and 1-80 mA, respectively, are explored. In planes parallel to the bottom wall, the flow displays an oscillating dipole vortex, while in planes normal to this wall shows an oscillatory boundary layer that resembles the oscillating Stokes layer. PIV measurements reveal that in each cycle vortices are created in the boundary layer in regions where the non-uniformity of the magnetic field is stronger. An approximate analytical solution of the oscillatory boundary layer, that considers the decay of the magnetic field in the normal direction, is obtained through a balance of viscous, inertial and electromagnetic forces. Additionally, a quasi-two dimensional numerical solution that reproduces the main features of the vortex flow, is presented.

Sunday, November 22, 2009 4:15PM - 6:25PM –

Session EU Granular III: Segregation and Mixing 200I

4:15PM EU.00001 Density Segregation of Granular Materials , ANURAG TRIPATHI, D.V. KHAKHAR, Indian Institute of Technology, Bombay — We have studied the segregation of spherical particles of same size with different densities flowing on a rough bumpy inclined surface. An 'effective temperature' (T_E) relating diffusivity and mobility of the particles, in analogy with Stokes-Einstein relation obtained from the fluctuation-dissipation theorem for jammed systems, has been defined for granular systems. Using the Stokes law for a sphere in a viscous medium, we obtain a simple expression for T_E as a function of diffusivity D and viscosity η. A balance of segregation and diffusion fluxes leads to prediction of f, the fraction of heavy particles in the flowing layer that depends on effective temperature apart from the density difference and inclination of the plane. The proposed theory has no fitting parameter and is able to predict the extent of segregation very well through out the layer for different mixtures, even for systems that are far away from jamming.

4:28PM EU.00002 The Role of Friction in the Segregation of Particles in a Chute Flow , FEDERICO HERNANDEZ-SANCHEZ, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — When a granular media, compound of particles with different properties, flows de-mixing of the constituent components may occur. This phenomenon, known as segregation, has been observed in particles with different sizes (the Brazil nut effect). In spite of being a well studied process, there is not a general understanding of the mechanisms that dominate this process. It is our interest to study the conditions that determine particle segregation in avalanches. Using a two-dimensional, discrete-element simulation, we study a bidisperse flow of particles over an inclined plane with periodic boundaries. We vary the roughness of the wall by attaching fixed particles on it. A wide range of parameters were varied (slope, number of particles, size ratio, density ratio). Both ordinary and reverse segregation patterns were observed. In this talk some preliminary results and their interpretation will be presented.

4:41PM EU.00003 Numerical analysis of the equilibrium behavior of two-phase granular mixtures , CHRISTOS VARSAKELIS, MILTIADIS PAPALEXANDRIS, Universite Catholique Louvain — In this talk we present a numerical analysis of the the equilibrium behavior of two-phase granular mixtures, as predicted by the model in *M.V. Papalexandris, J. Fl. Mech. (2004), 517, 103-112*. The equilibrium equations consist of an overdetermined system of quasi-linear partial differential equations with respect to the pressure and the volume fraction of the granular phase. Based on the Helmholtz decomposition and Ladyzhenskaya's decomposition theorem we develop a projection-type numerical method that overcomes the overdeterminacy of the system. The proposed method is proven to be both stable and consistent, hence, convergent. Further, it is general enough and can be applied to a variety of continuum models of complex, non-Newtonian mixtures. The talk concludes with the presentation of representative numerical results.

4:54PM EU.00004 Bulk horizontal size segregation in circular and parallel split-bottom cell[1] , YI FAN, KIMBERLY HILL, St. Anthony Falls Lab, Department of Civil Engineering, University of Minnesota — We perform Discrete Element Method (DEM) simulations of mixtures of different sized granular materials sheared in circular and parallel split- bottom cells. Horizontal segregation patterns in bulk are observed in both systems, but it appears that the underlying dominant driving mechanisms are different. In the curved cell, a global vertical convection roll is observed whose center is located at the plane of highest shear rate. The boundary between regions dominated by larger particles and those dominated by smaller particles are separated by the centers of the convection roll and the shear zone. In the parallel system, there is no such global convection. Nevertheless, a horizontal segregation pattern emerges, in which larger particles accumulate to the middle of shear zone and smaller particles migrate to the edges of shear zone. In this case the horizontal segregation may be caused by horizontal shear gradients and associated kinematic features.

[1]Acknowledge Support from NSF No. CMS-0625022 and NSF No. EEC-0630603.

5:07PM EU.00005 Dynamics of Large Intruder Particles in a Split Bottom Cell , ZHEN SUN, Department of Mechanical Engineering, University of Minnesota, YI FAN, KIMBERLY HILL, St. Anthony Falls Laboratoy, Department of Civil Engineering, University of Minnesota — We have performed experimental studies on the behavior of a single intruder particle in an otherwise relatively uniform matrix of granular materials sheared in a split bottom cell. We study the effect of the size and density of the intruder particle relative to the particles in the matrix on the behavior of the intruder particle. When an intruder particle is sufficiently large relative to the size of the particles in the matrix, it will rise to the surface only when the ratio between the density of the intruder particle and that of the matrix particle density d_r is somewhat less than a critical value $d_{cr1} < 1$. Intruder particles of a higher density move to an equilibrium distance h from the bottom of the cell that varies with d_r. When d_r is greater than a second critical value d_{cr2} – where $d_{cr1} < d_{cr2} < 1 - h = 0$. We model the behavior of these large intruder particles considering effective buoyancy and volume fraction variations as well as drag forces in the shear flow.

5:20PM EU.00006 Shear-segregation and mixing of sheared bidisperse granular materials , KAREN DANIELS, LAURA GOLICK, Dept. of Physics, NC State Univ., LINDSAY MAY, MICHAEL SHEARER, Dept. of Mathematics, NC State Univ. — We perform experiments on granular size-segregation in an annular Couette apparatus in which a layer of small particles mixes with, and then resegregates from, a layer of large particles beneath it. We model this process using a modification of the Gray-Thornton model in which we impose a nonlinear shear profile typical of boundary-driven, confined flows. The experimentally-measured exponential velocity profile provides an input to this one-dimensional nonlinear PDE and the resulting solution of the initial value problem is non-standard, involving curved characteristics. We further interpret these solutions by numerically connecting the segregation process to changes in packing fraction, and find qualitative agreement with experimental results. As in the experiment, mixing times are observed to be faster than segregation times. Interestingly, while the size-segregation of granular materials has generally been thought to proceed faster the greater the size difference of the particles we observe that the segregation rate is quite sensitive to both the particle-size ratio and the confining pressure on the system. As a result, we observe that particles of both dissimilar and similar sizes segregate more slowly than intermediate particle size ratios and interpret this anomalous behavior in terms of a species-dependent distribution of forces within the system.

5:33PM EU.00007 Particle-size segregation of granular materials under shear , MICHAEL SHEARER, LINDSAY MAY, NICK GIFFEN, KAREN DANIELS, North Carolina State University — Particle size segregation in avalanches occurs through shearing within the granular flow. In such a flow, large particles migrate upwards, their vacated spaces being filled by smaller particles. The Gray-Thornton continuum model is a scalar conservation law in two space variables and time, but with variable coefficients corresponding to the spatially dependent velocity in shear flow. Sharp interfaces separating different mixtures are shock wave solutions that typically form in finite time from smooth initial conditions. Shocks with more large particles below small are physically unstable, leading to time-dependent multidimensional patterns. An experiment in a Couette shear cell exhibits mixing and segregation predicted by theoretical solutions.

5:46PM EU.00008 Granular mixing in quasi-two-dimensional tumblers with a vanishing flowing layer , IVAN CHRISTOV, JULIO M. OTTINO, RICHARD M. LUEPTOW, Northwestern University — We study, numerically and analytically, a singular limit of granular tumbled flows in quasi-two-dimensional rotating drums. Focusing on two versions of the kinematic continuum model of such flows, we examine the transition to the limiting dynamics as the shear layer vanishes. The limiting behavior is shown to be the same for both versions of the continuum model. Moreover, we demonstrate that, just as in a three-dimensional spherical tumbler, the limiting no-shear-layer dynamical system belongs to a class of discrete discontinuous mappings called piecewise isometries. In doing so, we identify a new mechanism of mixing, in the absence of the usual streamline crossing mediated by the flowing layer, termed streamline jumping. This leads to complex, if not technically chaotic, dynamics as long as the tumbler and fill fraction are such that the free surface of the flow moves vertically and horizontally in time.

5:59PM EU.00009 Cutting and shuffling of a granular mixture in a spherical tumbler , GABRIEL JUAREZ, RICHARD M. LUEPTOW, JULIO M. OTTINO, Northwestern University, ROB STURMAN, University of Leeds, STEPHEN WIGGINS, University of Bristol — Good mixing in a fluid system is usually associated with chaotic advection. For a granular system, good mixing can be achieved through an entirely different mechanism that is well-known to mathematicians and card-players, 'cutting and shuffling,' which has theoretical foundations in a relatively new area of mathematics known as "piecewise isometries", PWIs. Cutting and shuffling experiments are conducted in a spherical tumbler of diameter D=14cm that is half-filled with two colors of d=1mm glass beads and can be rotated by arbitrary angles periodically about each of two horizontal, orthogonal axes. In order to connect experimental results, which have a finite thickness flowing layer, with theoretical PWI mappings, which have a zero-thickness flowing layer, a continuum model with a variable flowing layer depth is utilized. The PWI theory accurately predicts the experimental mixing in this granular system demonstrating that PWI theory captures the essential kinematic features responsible for the mixing of granular materials in a three-dimensional tumbler. Furthermore, PWI results in mixing without stretching, a characteristic of chaotic mixing.

6:12PM EU.00010 Coarsening in axial segregation, an entropic approach , MATTHIAS SCHRÖTER, MPI for Dynamics and Self-organization, TILO FINGER, RALF STANNARIUS, University Magdeburg — Binary mixtures in rotating drums can segregate into a stripe pattern where the individual stripes merge on long timescales [1]. Here we present an X-ray tomography study which indicates that the driving mechanism of this coarsening process might be the increase of configurational entropy [2].
[1] Finger *et al.* PRE **74** 031312 (2006)
[2] Edwards & Oakeshott, Physica A **157** 1080 (1989)

Sunday, November 22, 2009 4:15PM - 6:25PM –
Session EV Swimming I 205A-D

4:15PM EV.00001 Fluid Forces and Vortex Wakes of a Flapping Foil , TEIS SCHNIPPER, ANDERS ANDERSEN, TOMAS BOHR, Department of Physics and Center for Fluid Dynamics, Technical University of Denmark, JENS HONORÉ WALTHER, Department of Mechanical Engineering, Technical University of Denmark and Swiss Federal Institute of Technology Zurich, Chair of Computational Science — We present a combined numerical (particle vortex method) and experimental (soap film tunnel) study of a symmetric foil with pitching oscillations in a two-dimensional free stream. We vary the frequency and amplitude of the oscillations and observe von Kármán wake, inverted von Kármán wake, and wakes in which two vortex pairs form per oscillation period. We find a close correspondence between the numerically determined vortex structures and the thickness variations that visualize the flow in the soap film.[1] Numerically we obtain systematic maps with 25×40 simulations in the frequency and amplitude plane of both wake type and average forces and moments, and we discuss the drag-thrust transition in relation to the changes in wake structure. Finally, we investigate the time evolution of the fluid forces and its link to the vortex formation near the round leading edge and the vortex shedding at the sharp trailing edge.

[1]Schnipper, Andersen, and Bohr, J. Fluid Mech. **633**, 411–423 (2009).

4:28PM EV.00002 The Hydrodynamic Origin of Whale Flukeprints , RACHEL LEVY, Harvey Mudd College — Whales swimming at a shallow depth leave a signature on the ocean surface known as a whale flukeprint. The print is a large, smooth, oval patch surrounded by a small wake or ridge. Informal observations made by biologists have led to hypotheses that the prints are made either by hydrodynamic structures created by the motion of the fluke, or by surfactants. This study employs experiments with an artificial fluke to determine whether prints can be created by hydrodynamic forces without the presence of surfactant. The effect of swim velocity on the width, length and duration of a flukeprint created by the artificial fluke is discussed. The experimental data indicate that prints can be formed solely by hydrodynamic forces. This conclusion is supported by observations of whales, infrared images of footprints and numerical simulations of vorticity.

4:41PM EV.00003 Three-dimensional wake of a biologically-inspired propulsor[1] , MELISSA GREEN, CLARENCE ROWLEY, ALEXANDER SMITS, Princeton University — Digital Particle Image Velocimetry (DPIV) was used to investigate the wakes of rigid pitching panels with a trapezoidal planform geometry, chosen to model idealized fish caudal fins. Experiments were performed for Strouhal numbers of 0.17 and 0.23. The three-dimensional unsteady vortex wake downstream of the panel trailing edge was visualized using spatially- and temporally-resolved two-component data. A Lagrangian Coherent Structure (LCS) analysis was employed in addition to Eulerian vortex identification criteria to investigate the generation and evolution of the wake. A reverse von Kármán vortex street pattern was observed near the mid-span immediately downstream of the panel trailing edge, but the complexity and three-dimensionality of the wake increases away from the mid-span as streamwise vortices interact with the swept edges of the panel. Farther downstream of the trailing edge, the wake was observed to shrink in the spanwise direction at both Strouhal numbers. In addition, a quantitative bifurcation in the LCS coincided with a qualitative transition of the wake structure observed with increasing Strouhal number.

[1]Supported by NIH CNRS Grant 1R01NS054271 and ONR MURI Grant ONR N00014-08-1-0642.

4:54PM EV.00004 Multi-directional thrusting using oppositely traveling waves in knifefish swimming , OSCAR CURET, MALCOLM MACIVER, NEELESH PATANKAR, Department of Mechanical Engg., Northwestern University — *Apteronotus albifrons*, also known as the black ghost knifefish, generate a weak electric field for omnidirectional sensing. This is matched by an extraordinary multi-directional swimming ability that is achieved by undulating a ribbon-like anal fin. Forward or backward motion is generated by a traveling wave on the ribbon fin. We have discovered that, for hovering and vertical swimming, the knifefish use two oppositely traveling waves on the ribbon fin. To understand the hydrodynamic mechanism of hovering and heave we performed fully resolved simulations of self-propulsion of the knifefish. We used kinematic inputs based on experimental observations. We found that the counter propagating waves generate two opposite streamwise jets along the bottom edge of the ribbon fin. These two jets meet approximately at the mid-section along the fin length and are deflected downward. The resultant downward momentum imparted to the fluid creates an upward force on the fish body which can be used for hovering or vertical swimming. There is a vortex ring pair of opposite directions at the middle of the fin that is associated with this fluid flow. Further insight into how the knifefish control heave and hovering was obtained from the measurements of force generated by a robotic ribbon fin for different wave parameters.

5:07PM EV.00005 On the "momentum enhancement" and hydrodynamic efficiency of gymnotiform and balistiform swimmers , ANUP SHIRGAONKAR, Department of Mechanical Engg., MIT, NEELESH PATANKAR, MALCOLM MACIVER, Department of Mechanical Engg., Northwestern University — Gymnotiform and balistiform swimmers generate thrust by undulating ribbon fins while keeping the body nearly rigid. The question of whether there is a hydrodynamic basis for this evolutionary adaptation was considered by Lighthill and Blake. They used a two-dimensional inviscid approach and explained this adaptation based on their finding that the thrust produced by an undulatory ribbon fin is much higher when it is attached to a rigid body. This was termed momentum enhancement. We revisited this problem by performing high-resolution numerical simulations to calculate the thrust generated by undulatory ribbon fins in a plate-fin model of a gymnotiform swimmer. We did not find momentum enhancement. This disagreement could be explained by noting that an axial jet along the bottom edge of the ribbon fin is the primary thrust producing mechanism. This flow is not significantly affected by the presence of the body thus leading to no momentum enhancement. Lighthill's theory does not capture this dominant mechanism of thrust production. We find that the observed relative size of the body and the ribbon fins is such that it tends to optimize the cost of transport, as opposed to simply maximizing thrust. We present scaling analysis that supports this finding.

5:20PM EV.00006 Labriform swimming of a ray-strengthened pectoral fin[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. Pectoral fins of bony fishes usually consist of a soft collagen membrane strengthened by embedded flexible rays. Morphologically, each ray is connected to a group of muscles so that the fish can control the rotational motion of each ray individually, enabling multi-degree of freedom control over the fin motion and deformation. We have developed a fluid-structure interaction model to simulate the kinematics and dynamic performance of a structurally idealized fin. This method includes a boundary-element model of the fluid motion and a fully-nonlinear Euler-Bernoulli beam model of the embedded rays. Using this model we studied thrust generation and propulsion efficiency of the fin at different combinations of parameters. Effects of kinematic as well as structural properties are examined. It has been illustrated that the fish's capacity to control the motion of each individual ray, as well as the anisotropic deformability of the fin determined by distribution of the rays, are essential to high propulsion performance. Specifically, it is found that a reinforced ray at the leading edge leads to performance enhancement.

[1]This study is supported by NSF under Grant CBET-0844857.

5:33PM EV.00007 Hydrodynamic characteristics of sailfish and swordfish[1] , WOONG SAGONG, WOO-PYUNG JEON, HAECHEON CHOI, Seoul National University — The sailfish and swordfish are known as fastest sea animals, reaching their maximum speeds of more than 100km/h. Recently, Sagong *et al.* (2008, Phys. Fluids) investigated the role of V- shaped protrusions existing on the sailfish skin in the skin-friction reduction but those protrusions did not make a direct role in reducing drag. On the other hand, the long bill has been regarded as a device of reducing drag by separation delay through turbulence generation. In the present study, we investigate the hydrodynamic characteristics of sailfish and swordfish by installing the stuffed ones in a wind tunnel and measuring the drag on their bodies and boundary-layer velocities above the body surfaces. The drag coefficients of sailfish and swordfish are 0.0075 and 0.009 based on the free-stream velocity and wetted area, respectively. They are comparable to or smaller than those of other kinds of fish such as the dogfish, tuna and trout. Next, the role of bill on the drag is studied. The drag without bill or with an artificial short bill is lower than that with the original long bill, indicating that the bill does not reduce the drag at all. From the velocity measurement near the body surfaces, we found that flow separation does not occur even without bill, and thus the conjecture that the flow separation is delayed through turbulence generation by the bill is not valid.

[1]Supported by the National Research Laboratory Program, MEST.

5:46PM EV.00008 Turbulence augmentation to achieve separation control over a bristled shark skin model[1] , LEAH MENDELSON, Franklin W. Olin College of Engineering, AMY LANG, DREW SMITH, University of Alabama — The skin of fast-swimming sharks is covered with scale-like denticles capable of bristling to form cavities instead of lying flat against the body. These may be valuable for delaying flow separation and reducing net drag forces. This complex 3D roughness geometry alters the flow through the creation of vortices in the cavities and augments the behavior of the boundary layer. Understanding these flow phenomena is necessary for replicating the shark's passive flow control mechanisms. A model of bristled shortfin mako denticles in turbulent flow was tested in a water tunnel facility using Digital Particle Image Velocimetry (DPIV) to study the impact of the shark skin on the boundary layer. Results of these experiments, including influence on the time-averaged boundary layer profiles and Reynolds stresses over the span of the model, will be discussed.

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

5:59PM EV.00009 Effect of Varying the Angle of Attack of the Scales on a Biomimetic Shark Skin Model on Embedded Vortex Formation[1] , JENNIFER WHEELUS, AMY LANG, University of Alabama — The skin of fast-swimming sharks is proposed to have mechanisms to reduce drag and delay flow separation. The skin of fast-swimming sharks is covered with small denticles, on the order of 0.2 mm, that if bristled create cavities. It has been shown that for an angle of attack of 90 degrees, vortices form within these cavities and impose a partial slip condition at the surface of the cavity. This experiment focuses on smaller angles of attack for denticle bristling, closer to the range thought to be achieved on real shark skin. A 3-D bristled shark skin model with varying angle of attack, embedded below a boundary layer, was used to study the formation of cavity vortices through fluorescent dye visualization and Digital Particle Image Velocimetry (DPIV). The effect of varying angle of attack on vortex formation will be discussed.

[1]Funding for this research has been received from the Lindbergh Foundation, NASA AL-EPSCoR, and NSF.

6:12PM EV.00010 Energetics and Motion Planning for Hamiltonian Fishlike Locomotion[1] , SCOTT KELLY, PARTHESH PUJARI, University of North Carolina at Charlotte — The self-propulsion of an undulating body suspended in a fluid hinges on the judicious excitation of the fluid itself, in that forward momentum is developed by the body as equal and opposite momentum is imparted to the fluid. The manner in which this is accomplished is recorded in the structure of the body's wake; the efficiency with which it's accomplished is reflected in the fluid's evolving kinetic energy. Focusing on a model for the self-propulsion of a free hydrofoil with variable camber in an infinite planar fluid, we examine relationships between economy of deformation, wake structure, and wake energetics. The model in question simplifies the underlying physics by restricting vortex shedding to the trailing point of the foil, discretizing shed vorticity, and neglecting dissipation — allowing the remaining dynamics to be framed in a Hamiltonian setting. We consider the self-propulsion of the foil both in a quiescent fluid and in a variety of vortex flows representing wake structures present in vehicle schools.

[1]Supported in part by NSF grant CMMI 0822817.

Sunday, November 22, 2009 4:15PM - 6:25PM –

Session EW Mini-Symposium on Uncertainty Quantification in Simulations of Fluid Flow 208A-D

4:15PM EW.00001 Scattering of shock waves by random roughness , GEORGE E. KARNIADAKIS, Brown University — We employ second-order stochastic perturbation analysis and stochastic simulations synergistically to study supersonic flow past a rough wedge. The roughness is modeleed as stochastic process obtained from a new stochastic equation. We derive explicit formulas for the perturbed solution, which along with the stochastic simulation results aid to elucidate the physical scaling laws. For small roughness height and correlation length, the mean of the perturbed pressure scales quadratically with the height and the inverse correlation length while the corresponding variance scales linearly. Our results are useful in evaluating the effects of roughness in high-speed flight but also in designing novel enhanced-lift aerodynamic surfaces using rough skin concepts.

4:41PM EW.00002 Stochastic Dynamics Models for Laminar-to-Turbulent Transition on a Flight Vehicle , RICH V. FIELD, Sandia National Laboratories — Random pressure fluctuations within a turbulent boundary layer provide dynamic excitation to a flight vehicle. These random fluctuations must be adequately modeled and appropriately utilized by high fidelity structural dynamics finite element models in order to achieve accurate predictions of structural random vibration response. Most work in this area assumes the fluctuating pressure field to be fully turbulent. However, most flight vehicles experience a transition within the boundary layer from laminar to turbulent flow. This transition event also provides dynamic excitation to the vehicle, resulting in structural response that, for some systems, can be very significant. Herein, we provide a model for the pressure field acting on a flight vehicle in the laminar-to-turbulent transition zone for the purpose of predicting associated structural vibration response. The model is a Gaussian random field modulated by a Poisson field to capture the intermittent characteristics of turbulent flow in the transition zone. Computer algorithms to generate samples of the random pressure field are provided, and these samples are applied to a finite element model for a flexible beam with attached oscillator for illustration.

5:07PM EW.00003 Optimal design and uncertainty quantification in blood flow simulations for congenital heart disease , ALISON MARSDEN, University of California, San Diego — Recent work has demonstrated substantial progress in capabilities for patient-specific cardiovascular flow simulations. Recent advances include increasingly complex geometries, physiological flow conditions, and fluid structure interaction. However inputs to these simulations, including medical image data, catheter-derived pressures and material properties, can have significant uncertainties associated with them. For simulations to predict clinically useful and reliable output information, it is necessary to quantify the effects of input uncertainties on outputs of interest. In addition, blood flow simulation tools can now be efficiently coupled to shape optimization algorithms for surgery design applications, and these tools should incorporate uncertainty information. We present a unified framework to systematically and efficient account for uncertainties in simulations using adaptive stochastic collocation. In addition, we present a framework for derivative-free optimization of cardiovascular geometries, and layer these tools to perform optimization under uncertainty. These methods are demonstrated using simulations and surgery optimization to improve hemodynamics in pediatric cardiology applications.

5:33PM EW.00004 Calibration, Validation and Uncertainty Quantification for Hypersonic Reentry Vehicles[1] , R.D. MOSER[2], University of Texas at Austin — At the Center for Predictive Engineering and Computational Sciences at the University of Texas, we are engaged in an effort to characterize the uncertainties encountered in simulations of hypersonic reentry vehicles. Uncertainties arise from the modeling of several complex phenomena such as aerochemistry, thermal radiation, turbulence and ablation, and their interactions. Our approach to characterizing uncertainty is intimately connected to the process of determining uncertain model parameters through calibration and the assessment of model veracity through validation. The outlines of this approach will be described, including the use of Bayesian inference, treatment of uncertainty arising from modeling errors and validation criteria. As examples, application of this uncertainty quantification framework to several component problems will be described, including turbulence and chemical reaction mechanisms. Also discussed will be the sensitivities of simulations of a full-scale reentry vehicle to the various sources of uncertainties discussed above.

[1]This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].
[2]In collaboration with: T. A. Oliver, E. E. Prudencio, S. M. Prudhomme, University of Texas at Austin

5:59PM EW.00005 Wind Tunnel to Flight: Numerical Simulations of Hypersonic Propulsion Systems[1] , GIANLUCA IACCARINO[2], Stanford University — Uncertainties in the flight conditions and limitations of ground based facilities create inherent difficulties in assessing the performance of hypersonic propulsion systems. We use numerical simulations to investigate the correlation of wind-tunnel measurements (Steelant et al., 2006) and flight data (Hass et al., 2005) for the HyShot vehicle; the objective is to identify potential engine unstart events occurring under different combustion regimes. As a first step we perform simulations corresponding to both reacting and non-reacting conditions in the ground-based facility to validate the numerical tools. Next, we focus on reproducing the flight conditions; a fundamental difficulty is the lack of precise information about the vehicle trajectory. A Bayesian inversion strategy is used to infer the altitude, angle of attack and Mach number from the noisy pressure measurements collected during the flight. The estimated conditions, together with the scatter due to the measurement uncertainty, are then used to study the flow and thermal fields in the combustor. The details of the methods used to characterize the uncertainty in the flow simulations and to perform the Bayesian inversion will also be discussed.

[1]This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number NA28614.
[2]In collaboration with: G. Iaccarino, R. Pecnik, A. Doostan, V. Terrapon, Stanford University.

Sunday, November 22, 2009 7:00PM - 9:30PM –

Session F1 Reception Hilton Minneapolis Minneapolis Ballroom - Third Floor

7:00PM F1.00001 Reception —All paying attendees receive a ticket to this event. Come to the third floor of the Hilton Minneapolis to visit with your colleagues. This evening is a highlight of the conference each year and is a great venue to interact with others in the field.

Monday, November 23, 2009 8:00AM - 9:57AM –

Session GA Turbulent Boundary Layers: Experiments II 101A

8:00AM GA.00001 Measurements of spatio-temporal spectra in a zero-pressure gradient turbulent boundary layer , J. LEHEW, M. GUALA, B.J. MCKEON, Graduate Aerospace Laboratories, California Institute of Technology — The structure of the velocity field in wall-parallel planes in a zero pressure gradient turbulent boundary layer is interrogated using time-resolved digital particle image velocimetry at moderate Reynolds number ($Re_\tau = 500$). In order to investigate turbulent boundary layer structure in light of the emerging understanding of the nature of very large scale motions (VLSMs), two high speed cameras are placed side-by-side to recover streamwise structures over ten times the boundary layer thickness in length, while still resolving the dissipative scales. The simultaneous spatial (streamwise and spanwise) and temporal joint spectra and correlations are used to investigate the validity of Taylor's hypothesis as the wall is approached, in light of the known significant wall normal extent of the VLSMs and the associated range of convection velocities.

8:13AM GA.00002 Instantaneous shear stress distribution in a turbulent wall-bounded flow , OMID AMILI, JULIO SORIA, Laboratory for Turbulence Research in Aerospace and Combustion, Department of Mechanical and Aerospace Engineering, Monash University, Australia — Knowledge of wall shear stress is crucial for understanding of all wall-bounded turbulent flows and also for many technical applications. The aim of the present work is to develop a novel stress sensor which is capable of measuring surface shear stress over an extended region of the flow. This sensor as a direct method for measuring surface stresses consists of mounting a thin film made of a elastic polymer on the surface of the solid model. The geometry and mechanical properties of the elastomer are measured, particles acting as markers are applied on the film surface, and an optical technique is used to measure the film deformation caused by the flow. While the technique can be used in air or water, its sensitivity can be tuned for different flow conditions. The static and dynamic calibration of the sensor, and its application to a fully developed turbulent channel flow at moderately high Reynolds numbers will be addressed, and results will be compared with indirectly measured wall shear stress from PIV experiment.

8:26AM GA.00003 Time-resolved PIV in fully developed turbulent pipe flow , LEO HELLSTROEM, ALEXANDER SMITS, Princeton University — Stereoscopic particle image velocimetry was used to study the three component flowfield in both fully developed turbulent pipe flow and at several locations downstream of a horizontal 90° bend. The data was acquired with a high speed camera, making it possible to resolve the flow field in time for Reynolds numbers up to $35'000$. The secondary motions downstream the bend appears to be governed by either a Dean type motion were two swirls with opposite signs coexists, one in the upper and one in the lower half of the pipe respectively. Or a motion where two swirls, as large as the pipe, with opposite sings are alternating between each other. Both motions are present at the same Reynolds numbers, but the unsteady behavior appears to be more common for higher Reynolds numbers.

8:39AM GA.00004 3-D Flow Visualization of a Turbulent Boundary Layer[1] , BRIAN THUROW, STEVEN WILLIAMS, KYLE LYNCH, Auburn University — A recently developed 3-D flow visualization technique is used to visualize large-scale structures in a turbulent boundary layer. The technique is based on the scanning of a laser light sheet through the flow field similar to that of Delo and Smits (1997). High-speeds are possible using a recently developed MHz rate pulse burst laser system, an ultra-high-speed camera capable of 500,000 fps and a galvanometric scanning mirror yielding a total acquisition time of 136 microseconds for a 220 x 220 x 68 voxel image. In these experiments, smoke is seeded into the boundary layer formed on the wall of a low-speed wind tunnel. The boundary layer is approximately 1.5" thick at the imaging location with a free stream velocity of 24 ft/s yielding a Reynolds number of 18,000 based on boundary layer thickness. The 3-D image volume is approximately 4" x 4" x 4". Preliminary results using 3-D iso-surface visualizations show a collection of elongated large-scale structures inclined in the streamwise direction. The spanwise width of the structures, which are located in the outer region, is on the order of 25 – 50% of the boundary layer thickness.

[1]This work is partially supported by the Army Research Office.

8:52AM GA.00005 Harvesting energy from turbulence in boundary layers by using piezoelectric generators , YIANNIS ANDREOPOULOS, DOGUS H. AKAYDIN, NIELL ELVIN, City College of New York, Mechanical Engineering — The availability of significant kinetic energy in fluid flows distributed over a number of temporal and spatial scales creates a unique opportunity to convert this energy into electrical output by using piezoelectric generators. The unsteadiness due to turbulence can produce mechanical strain energy in the piezoelectric material which in turn can generate a build up of charge that can be used to power electronic devices. In the present work, short length piezoelectric beams were placed in a zero pressure gradient two dimensional turbulent boundary layer at Reynolds numbers based on momentum thickness up to 6500 to evaluate their performance as energy generators. The piezoelectric beam was traversed across the boundary layer to determine the location where the output power is maximized. It was found that the location of maximum power is not close to the wall where most of the turbulent activities are high but further away from the wall. The work has shown that there is a three-way coupled interaction between the fluid flow, the piezoelectric structure and its electromechanical field.

9:05AM GA.00006 Turbulent Coherent Structures in a Thermally Stable Boundary Layer , OWEN WILLIAMS, SEAN BAILEY, ALEXANDER SMITS, Princeton University — An experiment was conducted to examine the effect of thermal stability on turbulent coherent structures occurring in a flat plate boundary layer. The objective is to further characterize the turbulence in thermally stable atmospheric boundary layers, commonly found in the arctic regions, focusing on Reynolds number independent effects. This experiment was conducted in a 16 foot long, 4'x2' cross-section, open-return wind tunnel by replacing the upper surface with a heated half inch aluminum plate. The plate was maintained at an isothermal condition, the boundary layer along this surface was tripped and the tunnel run at the lowest speed possible, in order to maintain both a fully turbulent boundary layer and a large Richardson number. A wide range of stabilities were investigated, with Richardson numbers ranging from 0 to 0.5, covering both the weakly and strongly stable regimes. Using thermocouple temperature measurements and time resolved particle image velocimetry; an attempt was made to identify changes in coherent turbulent motions corresponding to changing flow stability. Additionally, an attempt was made to identify significant features of the turbulence that could be used to identify clearly delineating features of the weakly stable and strongly stable flow regimes.

9:18AM GA.00007 Buoyancy effects on large- and small-scale turbulent motions in the atmospheric surface layer during the transition through neutral stability[1] , MEREDITH METZGER, University of Utah — The present study examines how the transitory nature of the atmosphere, both on diurnal and meso timescales, affects the evolution of Very Large-Scale Motions in the Atmospheric Surface Layer (ASL) during the transition through neutral thermal stability. It is hypothesized that the finite time duration of the near-neutral period arrests the development of VLSMs in the ASL, compared to those expected in a canonical turbulent boundary layer having equivalent Reynolds number; and that, this, in turn, affects the structure of the small-scale turbulence by impeding inner-outer interactions. These scientific questions are addressed using simultaneously sampled hot-wire and sonic anemometry time series obtained during a field campaign in Utah's western desert. The pointwise data span a wall-normal distance between 1 mm and 30 m above the surface over a time period of several hours centered around neutral transition. Velocity spectra as a function of time (i.e., thermal stability) are shown as well as statistics associated with the turbulent bursting process. Results are compared against those obtained both at lower Reynolds number in the laboratory and in the neutral ASL.

[1]Supported by ONR (N00014-08-1-0866; R. Joslin, grant monitor).

9:31AM GA.00008 Turbulent structures in smooth and rough open channel flows , VESSELINA ROUSSINOVA, Research Associate, Department of Civil and Environmental Engineering, University of Windsor, RAM BALACHANDAR, Professor, Faculty of Engineering, University of Windsor — Turbulent flow in open channels is unique because it is bounded by the free surface and the flow is entirely dominated by the bed turbulence. Both experiments and simulations agree that the large-scale near-wall structures interact with the free surface without significant reduction of their strength. Still the link between bed and free-surface turbulence is not well understood and it is of a particular importance for the processes occurring at the surface in shallow geophysical flows. In this paper, high resolution particle image velocimetry (PIV) measurements in an open channel flow are presented. Velocity measurements were obtained in the streamwise - wall normal (x-y) plane and streamwise-spanwise (x-z) plane. Streamwise-spanwise (x-z) planes were acquired at various vertical locations. The focus of this study is to investigate the streamwise oriented vortices along the flow depth as documented through the analysis of swirling strength, conditional averaging and vortex statistics. Turbulence measurements in rough open channel flow are also presented and compared with those on the smooth wall.

9:44AM GA.00009 A turbulent boundary layer on a rough wall at hypersonic speeds , DIPANKAR SAHOO, PARTHAV DESAI, ALEXANDER SMITS, Princeton University — Previous experiments on hypersonic turbulent boundary layers have documented the general features of the mean flow behavior on a smooth plate, but virtually no data exist describing the boundary layer behavior on a rough wall for Mach numbers greater than about 5. Here, we report PIV measurements of the mean flow and two components of velocity fluctuations on a flat plate with three different roughness geometries: a square bar roughness, and two diamond roughness elements of different height. The boundary layer develops at Mach 7.2 in a perfect gas, at a Reynolds number based on momentum thickness of about 3600. The results are compared with DNS under identical flow conditions. Supported under NASA Grant NNX08AB46A, Program Manager Catherine McGinley.

Monday, November 23, 2009 8:00AM - 9:57AM –

Session GB Turbulence: Fundamentals I 101B

8:00AM GB.00001 The bottleneck effect and the Kolmogorov constant in three-dimensional turbulence[1] , DIEGO DONZIS, Texas A&M University, K.R. SREENIVASAN, ICTP, Italy; U. Maryland — A large database generated from direct numerical simulations (DNS) of isotropic turbulence, including recent simulations at up to 4096^3 resolution and Taylor microscale Reynolds numbers of up to about 1000, is used to explore the bottleneck effect in three-dimensional energy spectrum and in second-order structure functions, and to determine the Kolmogorov constant, C_K. The difficulties in estimating C_K at any finite Reynolds number are examined. Our data from well-resolved simulations show that the bottleneck effect decreases with the Reynolds number and that its behavior is independent of the nature of the forcing scheme and is insensitive to small-scale resolution. This trend is seen in both spectral and physical spaces, though the effect is less noticeable in the latter. An alternative to the usual procedure for determining C_K is suggested. The proposal does not depend on a particular choice of fitting ranges or power-law behavior in the inertial range. Within the resolution of the numerical data, C_K thus determined is constant in the Reynolds number. A simple model including non-local energy transfer is proposed to reproduce the observed scaling. Further implications of the findings are discussed.

[1]Supported by NSF Grant CBET-0553602.

8:13AM GB.00002 Split energy cascade in turbulent flows , DARIO VINCENZI, CNRS UMR 6621, Laboratoire J.-A. Dieudonne, Universite de Nice Sophia Antipolis, ANTONIO CELANI, Institut Pasteur, Paris, France, STEFANO MUSACCHIO, Dipartimento di Fisica Generale, Università di Torino, Italy, SYLVAIN RUBENTHALER, Laboratoire J.-A. Dieudonne, Universite de Nice Sophia Antipolis, France — Hydrodynamic turbulence exhibits a remarkable dependence on the space dimension. This property manifests for instance in the direction of the kinetic-energy cascade, which is direct in 3D and inverse in 2D. A passive scalar transported by a turbulent flow shows an analogous behavior. In isotropic flows, the variance of the scalar field cascades either downwards or upwards depending on the dimension, the degree of compressibility, and the scaling exponent of the carrier flow. We undertake a geometrical approach to investigate the dependence of turbulence and turbulent transport on the space dimensionality. We first consider a system that is fully under analytical control, i.e. a passive scalar transported by a Gaussian short-correlated flow on a cylindrical surface, where the radius can be inflated or collapsed at will. For any finite radius, the variance cascade splits into a direct branch and an inverse one. This behavior is intimately connected to the existence of a non-degenerate invariant measure for the fluid-particle separations. Direct numerical simulations of the Navier-Stokes equations show that also the kinetic-energy cascade splits when the aspect ratio of the flow is less than one.

8:26AM GB.00003 Controlling the Dual Cascade of Two-dimensional Turbulence , MOHAMMAD FARAZMAND, NICHOLAS KEVLAHAN, McMaster University — It has been established that monoscale forcing cannot produce the dual cascades of energy and enstrophy with the scaling laws predicted by Kraichnan–Leith–Batchelor (KLB) theory. However, we have been able to find forcings which do produce the KLB scalings: $E(k) \propto k^{-5/3}$ for the inverse energy cascade and $E(k) \propto k^{-3}$ for the forward enstrophy cascade. We find these forcings using a novel adjoint-equation-based optimal control technique. First, the control problem is formulated and a method for controlling the energy spectrum of solutions of the incompressible Navier–Stokes equations is proposed. The control process is validated by several test cases. Then, this control method is applied to a pseudo-spectral numerical computation of the 2-D incompressible Navier-Stokes equations with doubly periodic boundary conditions in order to find the forcing that reproduces the scaling laws of KLB theory. Finally, we demonstrate that the flows we obtain are indeed dynamically active by measuring directly the energy and enstrophy fluxes. We also compare our forcing and the resulting turbulence with results obtained using a linear forcing recently proposed by Lundgren (2003). The results presented here show that the choice of forcing can fundamentally alter the dynamics and spectral properties of the turbulence, and that the theoretically attractive choice of band-width limited forcing is actually inconsistent with KLB theory.

8:39AM GB.00004 Forward/Inverse Energy Cascade in 2D and QG Turbulence , CHUNG-HSIANG JIANG, PHILIP S. MARCUS, University of California, Berkeley — We perform numerical simulations to study two-dimensional and quasi-geostrophic turbulence. In all runs, small scale forcing injects energy at wave number, k_f, and the inverse energy cascade is halted at large scale by linear drag. A new decomposition of spectral energy flux into aggregated transfer function cascading up and down (hereafter ATFu and ATFd) is introduced instead of Kraichnan's classical approach. Both functions are positive, monotonically increasing in spectral space and have the same power-law dependency in the energy inertial range. Amazingly, the ATFd has discontinuity at k_f and the jump equals to energy injection rate $\dot{E}_{in}$ while the ATFu is always continuous. This implies that the energy injected is transferred first to small scale and then cascade inversely but not directly to large scale. The QG turbulence resembles the 2D turbulence if β effect is too weak to create zonal flow. If zonal jets are spotted, the exponent of the power-law dependency and the magnitude of ATFs are smaller compared to that in 2D turbulence indicates that β inhibits the inverse energy cascade. The properties of ATFs are strongly dependent on $\dot{E}_{in}$ and the drag loss but not k_f.

8:52AM GB.00005 Inverse Cascades and Zonal Flows on a Beta Plane , PHILIP MARCUS, CHUNG-HSIANG JIANG, University of CA at Berkeley — We examine the role of forward and inverse cascades in 2D turbulence in creating zonal jets on a β-plane. The magnitude of the characteristic velocity and the characteristic width of a zonal jet are set by the balance of the two cascades. The widths of the jets are strongly dependent on the value of the local Rossby deformation radius L_R. Kinetic energy is dominated by potential energy at length scales greater than $2\pi L_R$. We find that little energy inverse cascades to scales greater than $2\pi L_R$, and there is a break in the slope of the kinetic energy spectrum at that scale. Forcing at small scales produces large-scale zonal flows that resemble the widths, but not the magnitudes, of jet streams of Jupiter and Saturn. The magnitudes of the large-scale velocities of the computed zonal flows are much smaller than on Jupiter or Saturn. The transfer of energy from small scales to large scales involves many more wave number triads on an f-plane than on a β- plane, so the equilibrium energy of the large scale zonal flows is determined by only a few triads.

9:05AM GB.00006 What is turbulence and which way does it cascade? , CARL H. GIBSON, University of California at San Diego — Turbulence is defined as an eddy-like state of fluid motion where the inertial vortex forces of the eddies are larger than any of the other forces that tend to damp the eddies out. Inertial vortex forces vxw are zero for irrotational flows, so irrotational flows are not turbulent by definition even though they may be random and induced by turbulence. Because the vorticity w is always produced at small scales, turbulence always cascades from small scales to large. Turbulence growth is limited by vertical buoyancy forces at the Ozmidov scale of fossilization and by horizontal Coriolis forces at the Hopfinger scale of fossilization. Fossil turbulence is defined as a perturbation in any hydrophysical field produced by turbulence that persists after the fluid is no longer turbulent at the scale of the perturbation. Most turbulent mixing in the ocean and atmosphere occurs in fossil turbulence patches where most of the turbulent kinetic energy of the patch has radiated near vertically as fossil turbulence waves. Vertical heat, mass, momentum and information transport in the ocean is dominated by an intermittent generic process termed beamed zombie turbulence maser action mixing chimneys (see http: //maeresearch.ucsd.edu /~cgibson /Documents2007 /GibsonBB08Nov26_Alist.htm).

9:18AM GB.00007 Scale-locality of the energy cascade in turbulence using Fourier Analysis[1] , HUSSEIN ALUIE, GREGORY L. EYINK, Johns Hopkins University — We investigate the scale-locality of non-linear interactions which drive the energy cascade in a turbulent flow. The main picture that emerges from our work is that the primary participants in the cascade process are triplets of "eddies" comprised of adjacent *logarithmic bands* of Fourier modes. We disprove in particular an alternate picture of "local transfer by nonlocal triads" by showing that such triads, due to their restricted number, make a vanishingly small contribution to the energy flux in the inertial range. We rigorously prove that it is only the aggregate effect of a geometrically increasing number of local wavenumber triads which can sustain the energy cascade to small scales. Our analysis shows that the SGS definition of the flux is the proper measure of the cascading energy and that the sharp spectral filter has a firm theoretical basis for use in LES modeling. It also demonstrates the danger in the widespread notion that the elementary interactions in turbulence are those involving triads of single Fourier modes. We support our results with numerical data from a 512^3 pseudo-spectral simulation of isotropic turbulence with phase-shift dealiasing.

[1]Computer time provided by the Digital Laboratory for Multi-Scale Science at the Johns Hopkins University and support from NSF grant # ASE-0428325 are gratefully acknowledged.

9:31AM GB.00008 Quantifying the locality of nonlinear interactions in MHD turbulence , J.A. DOMARADZKI, University of Southern California, B. TEACA, D. CARATI, Universite Libre Bruxelles — The locality functions introduced by Kraichnan give the fraction of the energy flux across a given cutoff wavenumber k_c that is due to nonlinear interactions with wavenumbers k smaller than the cutoff (the infrared locality function) or greater than the cutoff (the ultraviolet locality function). Previous analysis of DNS data for hydrodynamic turbulence confirmed the theoretical scaling exponent of $n = 4/3$ in the wavenumber ratio and in the limit of the infinite inertial range. We have extended the analysis to DNS data for MHD turbulence. Out of four nonlinear terms contributing to the energy transfer, two dominant ones, $b \cdot \nabla b$ and $b \cdot \nabla u$, lead to the locality functions that exhibit behavior that can be characterized by scaling exponents in the infrared. The extend of the inertial range is insufficient to determine the exponents uniquely but the data are indicative of values between 1/2 and 2/3, i.e., much less than for hydrodynamic turbulence. Therefore, the nonlinear energy transfer is significantly more nonlocal in MHD turbulence, with potential implications for theory and modeling.

9:44AM GB.00009 Energy flux in non-equilibrium energy spectra in steady turbulence , KIYOSI HORIUTI, KENSAKU SAITOU, Tokyo Institute of Technology — The energy spectrum $E(k)$ and energy flux function $\Pi(k)$ in non-equilibrium state are obtained using the spectral energy equation based on the Kovasznay, Leith diffusion and Heisenberg hypothesis. The derived models are assessed using the DNS data for forced homogeneous isotropic turbulence. Three different forcing schemes are used and compared. In all these forcing schemes, the base spectrum obeys the Kolmogorov law $E(k) \propto k^{-5/3}$, and $\Pi(k) = \text{const}$, but the temporal development of the the deviatoric spectrum and flux is divided into the three phases. In the period in which $d\varepsilon/dt \equiv \dot{\varepsilon} > 0$, $E(k) \propto k^{-7/3}$ and $\Pi(k) \propto k^{-2/3}$ in the inertial subrange (Phase 1), while $E(k) \propto -k^{-7/3}$ and $\Pi(k) \propto -k^{-2/3}$ when $\dot{\varepsilon} < 0$ (Phase 2), where ε is the dissipation rate. In the transient period between Phase 1 and Phase 2, $\dot{\varepsilon} \approx 0$ and $\ddot{\varepsilon}$ is large, and $E(k) \propto k^{-9/3}$ and $\Pi(k) \propto k^{-4/3}$ (Phase T). On average, the deviatric spectrum induces the forward scatter of the energy into the small scale in Phase 1, and the backward scatter of the small scale energy into the large scale in Phase 2. These results are overall consistent with the prediction obtained using the closure models, but the eddy-viscosity Heisenberg model does not yields $E(k) \propto k^{-9/3}$ in Phase T. Due to the effect of the intermittency, the energy spectrum and flux exhibit a slight deviation in the exponents.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GC Turbulence Simulations IV 101C

8:00AM GC.00001 Study of local isotropy in a turbulent pipe flow using longitudinal and transverse structure functions , KHADIJA HMOUDOU, XIAOHUA WU, Royal Military College of Canada — The scaling exponents of the longitudinal $\langle \Delta u_z^n \rangle$ and the two transverse structure functions, $\langle \Delta u_r^n \rangle$ and $\left\langle \Delta u_\theta^n \right\rangle$ with $n \leq 7$ are studied in a fully developed incompressible turbulent pipe flow at $Re_D = 24580$ and 50000 using direct numerical simulation flow fields. The scaling exponents for $\langle \Delta u_r^n \rangle$ and for $\left\langle \Delta u_\theta^n \right\rangle$ increase with the turbulent Reynolds number R_λ. However, the scaling exponents for $\langle \Delta u_z^n \rangle$ remain nearly unchanged. The Kolmogorov universal constants in both of the dissipative range and inertial range for the longitudinal structure functions show a smaller increase with R_λ than those for the transverse structure functions. The present results are compared with previous experimental and DNS data for channel and duct flows (Antonia *et al.* (1997). Phys. Fluids, 9 (11), 3465)

8:13AM GC.00002 Tensor-based Lagrangian time correlations in DNS of isotropic turbulence[1] , HUIDAN YU, Johns Hopkins University, CHARLES MENEVEAU, Department of Mechanical Engineering and Institute for Data Intensive Engineering and Science, Johns Hopkins University — We study Lagrangian statistics of dynamically important tensors, such as velocity- gradient tensor, together with its symmetric and antisymmetric parts, through fluid particle tracking. The data, a 1024^4 space-time DNS of forced isotropic turbulence, are accessed using the web-services of the JHU public database (http://turbulence.pha.jhu.edu). A Tensor-based time-correlation function is defined by the tensor product between variables at different times along the Lagrangian trajectory. Analyses in the literature had shown slightly longer correlation times for the square rotation rate as compared to the square strain-rate magnitude. However, here we show that the difference is much larger when considering the dynamically more relevant tensor-based correlation function. The question whether these trends are due mostly to vortical coherent structures (worms) is addressed using conditional averaging. Even with the exclusion of worms, rotation-rate remains significantly more correlated over time than the strain-rate. The analysis is done for the pressure Hessian tensor and significant differences are obtained for its trace and deviatoric parts.

[1]Supported by the Keck Foundation and NSF (ITR).

8:26AM GC.00003 Pressure, acceleration and velocity structure functions in DNS at high Reynolds number and/or improved small-scale resolution[1] , K.P. IYER, P.K. YEUNG, Georgia Tech, R.J. HILL, Univ. of Colorado and NOAA. — Evidence from both numerical simulation and experiment in the literature suggest the second-order structure function of pressure fluctuations requires higher Reynolds number than the pressure spectrum for inertial range scaling. We use a direct numerical simulation database for isotropic turbulence at resolution up to 4096^3 to directly calculate the pressure structure function and pressure gradient two-point correlation. We examine the relationships between those statistics and those same quantities calculated from fourth-order velocity structure functions. Our results suggest that the longitudinal, mixed and transverse fourth-order velocity structure functions, usually denoted by $D_{LLLL}(r)$, $D_{LLNN}(r)$ and $D_{NNNN}(r)$, obey, to a good approximation, the same scalings for scale size r in the inertial range. While mutual cancellations limit the accuracy with which these can be used to evaluate the pressure structure function the ambiguities clearly become smaller at higher Reynolds number. We also use new datasets of improved small-scale resolution albeit at lower Reynolds number to re-examine the nature of pressure gradient and viscous force correlations at small scale separations, more definitively than possible before.

[1]Supported by NSF Grant CBET-0553867 (and IREE Supplement).

8:39AM GC.00004 Multi-particle Lagrangian statistics of turbulent dispersion from simulations of isotropic turbulence at $R_\lambda \approx 1100$[1] , J.F. HACKL, P.K. YEUNG, Georgia Institute of Technology, B.L. SAWFORD, Monash University — Numerical simulations at up to 4096^3 grid resolution have been conducted on machines with very large processor counts to obtain the statistics of Lagrangian particle pairs and tetrads in turbulent relative dispersion. Richardson-Obukhov scaling for mean-square pair separation adjusted for initial conditions is observed for intermediate initial separations, in support of prior estimates of about 0.6 for Richardson's constant. Simulations at $R_\lambda \approx 650$ have also been conducted for sufficient duration to obtain fully converged exit time statistics for independently moving particles at very large scales. The fact that all particle pairs reach such large scales of separation means the inertial subrange of exit times is also captured accurately. The results show Kolmogorov scaling for positive moments of exit time, but a strong dependence on initial separations for inverse moments. Inertial-range estimates of tetrad shape factors are reinforced by simulations at Taylor-scale Reynolds numbers up to about 1100. Tetrad shape parameters conditioned on cluster size are also examined in order to understand geometric features of turbulent dispersion in more detail.

[1]Supported by NSF Grant CBET-0553867 and advanced computing resources at NICS (NSF) and NCCS (DOE).

8:52AM GC.00005 DNS of the thermal effects of laser energy deposition in isotropic turbulence[1] , SHANKAR GHOSH, KRISHNAN MAHESH, University of Minnesota — Laser energy deposition in isotropic turbulence is studied using DNS. A spectral numerical method is combined with shock-capturing and numerical challenges faced are discussed. A model problem involving energy deposition near a single vortex is studied as a first step. For the turbulent problem, $Re_\lambda = 30$ and $M_t = 0.001$ and 0.3 are considered. Evolution of the mean flow is divided into shock formation, shock propagation and core roll up stages. For $M_t = 0.3$, the turbulence slows down shock formation and propagation and prevents core roll up. This behavior is not observed for $M_t = 0.001$. The turbulence intensities are enhanced due to compression from the shock wave and suppressed due to expansion in the core. Turbulent kinetic energy budgets are computed to explain this behavior. Effect of mean vorticity production on the turbulence is also studied. As an application, laser energy deposition near a wall is studied. Orientation of the laser axis and distance of the focal volume from the wall are found to affect the evolution of the resulting flow field.

[1]Project supported by Air Force Office of Scientific Research.

9:05AM GC.00006 Decomposition of Fluid Acceleration by Rotational and Irrotational Motion in Isotropic Turbulence , YONGNAM PARK, CHANGHOON LEE, Yonsei University — It is well known that fluid acceleration in turbulence is highly intermittent. Source of the intermittency was found to be closely related to the rotational motion of coherent vortical structures. From the Poisson equation for pressure, $\frac{1}{\rho}\nabla^2 P = \Omega - \frac{\epsilon}{2\nu}$, acceleration, which is mostly the negative of pressure gradient, can be expressed as a sum of acceleration-like terms, $-\nabla(\nabla^2)^{-1}\Omega + \nabla(\nabla^2)^{-1}\frac{\epsilon}{2\nu}$, each of which is named as a^Ω and a^ϵ . They are acceleration due to rotational motion of eddy and acceleration due to irrotational strain field, respectively. We investigated the statistical characteristics of those accelerations by using direct numerical simulation of isotropic turbulence. Flatness of acceleration is of order of 10 but flatness of a^Ω and a^ϵ are $3 \sim 5$ which represents less intermittency in the range of $Re_\lambda = 47 \sim 130$. Based on the cylinder vortex model, we show that probability density function of acceleration must have -5/3 slope and pdf's of a^Ω and a^ϵ must have -3 slope in log-log scale when the Reynolds number is infinite. Numerical and experimental results do not show clear slope since the Reynolds number is relatively low, but an asymptotic behavior is observed.

9:18AM GC.00007 Evolution of Compressible Decaying Isotropic Turbulence with Multi-temperature Non-equilibrium , WEI LIAO, YAN PENG, LI-SHI LUO, Old Dominion University — Understanding and predicting of transition and turbulence under non-thermodynamical-equilibrium (NTE) conditions are important for hypersonic flight and other industrial applications. In NTE turbulence, the Kolmogorov paradigm, which forms the basis of most equilibrium turbulence models, may be invalid. Furthermore, under the NTE conditions, multiple temperatures often take place in diatomic gases even at room temperature due to the insufficient particle collisions. Therefore, the effect due to the internal degrees of freedom interactions on turbulence physics is essential in non-equilibrium flows. Here, we apply gas kinetic scheme for DNS of compressible decaying isotropic turbulence with multi-temperature non-equilibrium. Our results show that the rotational collision number in the rotational energy relaxation model and the initial energy ratio of rotational and translational modes can significantly affect the evolution of the decaying turbulence.

9:31AM GC.00008 Large-eddy simulation of compressible flow over a backward-facing step using a spectral multidomain method , HARISH KANCHI, University of Illinois at Chicago, KAUSTAV SENGUPTA, Boeing Research and Technology-India, GUSTAAF JACOBS, San Diego State University, FARZAD MASHAYEK, University of Illinois at Chicago — Analysis of compressibility effects on separated curved shear layers in practical configurations has received little attention in the turbulence community. In this work, we perform large-eddy simulation (LES) of cold flow in an asymmetric dump-combustor with a spectral multi-domain method. The LES method combines a high-order multi-domain approximation with a dynamic sub-grid model and explicit interpolant-projection filtering to facilitate simulation at high Reynolds numbers. The inflow turbulence is modeled using a novel stochastic model, which is both efficient and general. We investigate the impact of the important physical parameters, such as the state of the boundary layer at separation, Reynolds number and Mach number as well as the interplay between them. One of the principal findings is the different responses of the transitional and turbulent shear layers with increase in compressibility. Increase in compressibility for the transitional flow causes a larger production of turbulent kinetic energy resulting in a faster growth of the shear layer. While for the turbulent shear layer, the growth rate is inhibited with increase in compressibility as a result of higher pressure-dilatation.

9:44AM GC.00009 The scaling of polymer drag reduction with polymer and flow parameters in turbulent channel flow , DONG-HYUN LEE, RAYHANEH AKHAVAN, Dept. of Mechanical Engineering, University of Michigan, Ann Arbor, MI 48109-2125, USA — The scaling of polymer drag reduction with polymer and flow parameters is investigated using results from direct numerical simulations (DNS) of dilute, homogeneous polymer solutions in a turbulent channel flow performed at a base Reynolds number of $Re_{\tau_b} \approx 230$. The full range of drag reduction from onset to maximum drag reduction (MDR) is reproduced in DNS with realistic polymer parameters, with results in good agreement with available experimental data. Onset of drag reduction is found to be a function of both the polymer concentration and the Weissenberg number (We_τ), in agreement with the predictions of DeGennes (1986). Saturation of drag reduction is achieved at a viscosity ratio of $\beta \approx 0.98$ at all We_τ, with the magnitude of drag reduction at saturation being a strong function of We_τ. A $We_{\tau_b} \sim O(Re_{\tau_b}/2)$ is needed to reach MDR. The presence of the polymer results in attenuation of turbulence at certain turbulent scales, determined by the Weissenberg number and the concentration. At saturation concentrations, the size of the largest attenuated eddy conforms to the predictions of Lumley (1969), while at concentrations below saturation, it conforms to a modified form of DeGennes (1986) theory. A mechanism of polymer drag reduction consistent with these observations will be presented.

9:57AM GC.00010 Turbulent Channel Flow With Λ Shape Turbulators on One Wall , JAIME TORO, BENJAMIN CRUZ, STEFANO LEONARDI, University of Puerto Rico at Mayaguez — tudy of turbulent heat or mass transport is of special interest in engineering, especially for heat exchangers. For instance, roughness elements (turbulators) are usually placed on the walls of the internal channels of a turbine blade to enhance the heat transfer. In the present paper, DNSs are carried out for passive heat transport in a turbulent channel flow with Λ shape square ribs for w/k = 1, 3, 7, 15 (w being the pitch, k the height of the ribs turbulators. The angle of inclination of the lambda shape turbulators is 45 degrees. Numerical results show that Λ shape square ribs are more efficient than square ribs in maximizing the heat transfer. The configuration with w/k=3 presents the largest heat flux. The increase in the heat transfer is due to a secondary motion which is generated by the Λ shape turbulators. Two counter rotating vortices above the square ribs transport the heat out of the wall into the center of the channel. The distribution of the heat flux coefficient is not uniform in the channel and leads to temperature gradients at the wall. The total drag of the Λ shape turbulators is larger than that over a smooth wall due to an increase of form drag.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GD Flow Control IV 101D

8:00AM GD.00001 Effect of Dielectric Properties on Functional Relationship between Plasma Initiation and Ambient Pressure , JOSEPH VALERIOTI, THOMAS CORKE, University of Notre Dame — A parameter study is conducted using Single Dielectric Barrier Discharge (SDBD) plasma actuators. An experimental setup is used to determine the plasma initiation voltage for a range of pressure, frequency, dielectric thickness, and dielectric material. An actuator is placed in a sealed chamber evacuated to a given negative gage pressure. Peak-to-peak voltage of an AC sine wave is then increased until plasma formation is verified by means of light intensity. The variation of this initiation voltage is determined as a function of ambient pressure for different combinations of thickness/material of the SDBD dielectric. The relationship is then presented so as to assess its collapsibility to a parameter describing the ratio of the dielectric coefficient to the dielectric thickness, which is central to lumped element models used in plasma modeling. Future work will examine the relationship at pressures above atmospheric.

8:13AM GD.00002 Effect of Actuation Parameters on Opposition Control of Transient Growth in a Blasius Boundary Layer Using Plasma Actuators , RONALD HANSON, PHILIPPE LAVOIE, University of Toronto, Aerospace Studies, AHMED NAGUIB, Michigan State University, Mechanical Engineering — This work is concerned with investigating an actuation scheme, using plasma actuators, designed to negate the effect of the transient growth instability occurring in a Blasius boundary layer and, by this means, delaying the bypass transition process. The actuators investigated consists of a spanwise array of symmetric plasma actuators, which are capable of generating spanwise periodic counter-rotating vortices. The effectiveness of the actuator array was tested on disturbances introduced via an array of cylindrical roughness elements. Early investigations have demonstrated a reduction of the total disturbance energy produced by the roughness elements by up to 68% depending on the actuator geometry. In the present study, the focus is on determining the effect that the excitation signal supplied to the actuator, such as waveform, frequency and amplitude, can have on the receptivity of the boundary layer to the plasma forcing. It is found that the excitation can change the modal content of the disturbance introduced inside the boundary layer by the actuator. The consequences of these results are discussed with respect to actuator modelling and issues related to eventual integration with a feedback control system for transition control.

8:26AM GD.00003 Preliminary numerical assessment of turbulent skin friction control with plasma actuators , BETTINA FROHNAPFEL, ANTONIO CRISCIONE, CAMERON TROPEA, Technical University Darmstadt, YOSUKE HASEGAWA, NOBUHIDE KASAGI, The University of Tokyo — Plasma actuators (PA) introduce a body force in the near-wall region of a fluid flow. This body force has already been successfully used for separation and transition flow control. We investigate the possibility of applying PAs to turbulent skin friction drag reduction by testing the effect of a modelled PA's body force in a numerically simulated turbulent channel flow. The body force is implemented into a control loop, which aims at impeding the spanwise velocity component near the wall surface. We assume to employ distributed sensors and actuators of finite size in order to investigate optimum actuator sizes for practical applications. Since the detailed physics of the body force generation by PAs and the resulting force distributions are still under study and a matter of discussion, we employ different models for the force distribution with the goal to identify the critical requirements for skin friction drag reduction with PAs.

8:39AM GD.00004 S-Duct Engine Inlet Flow Control Using SDBD Plasma Streamwise Vortex Generators[1] , CHRISTOPHER KELLEY, CHUAN HE, THOMAS CORKE, University of Notre Dame — The results of a numerical simulation and experiment characterizing the performance of plasma streamwise vortex generators in controlling separation and secondary flow within a serpentine, diffusing duct are presented. A no flow control case is first run to check agreement of location of separation, development of secondary flow, and total pressure recovery between the experiment and numerical results. Upon validation, passive vane-type vortex generators and plasma streamwise vortex generators are implemented to increase total pressure recovery and reduce flow distortion at the aerodynamic interface plane: the exit of the S-duct. Total pressure recovery is found experimentally with a pitot probe rake assembly at the aerodynamic interface plane. Stagnation pressure distortion descriptors are also presented to show the performance increase with plasma streamwise vortex generators in comparison to the baseline no flow control case. These performance parameters show that streamwise plasma vortex generators are an effective alternative to vane-type vortex generators in total pressure recovery and total pressure distortion reduction in S-duct inlets.

[1]Supported by Boeing, Inc.

8:52AM GD.00005 Development of plasma streamwise vortex generators for increased boundary layer control authority[1] , PATRICK BOWLES, DAVID SCHATZMAN, THOMAS CORKE, FLINT THOMAS, University of Notre Dame — This experimental study focuses on active boundary layer flow control utilizing streamwise vorticity produced by a single dielectric barrier discharge plasma actuator. A novel plasma streamwise vortex generator (PSVG) layout is presented that mimics the passive flow control characteristics of the trapezoidal vane vortex generator. The PSVG consists of a common insulated electrode and multiple, exposed streamwise oriented electrodes used to produce counter-rotating vortical structures. Smoke and oil surface visualization of boundary layer flow over a flat plate compare the characteristics of passive control techniques and different PSVG designs. Passive and active control over a generic wall-mounted hump model, Re_c = 288,000-575,000, are compared through static wall pressure measurements along the model's centerline. Different geometric effects of the PSVG electrode configuration were investigated. PSVG's with triangular exposed electrodes outperformed ordinary PSVG's under certain circumstances. The electrode arrangement produced flow control mechanisms and effectiveness similar to the passive trapezoidal vane vortex generators.

[1]Supported by Bell Helicopter

9:05AM GD.00006 Plasma Enhanced Aerodynamics of Wind Turbine Blades[1] , JOHN COONEY, THOMAS CORKE, ROBERT NELSON, University of Notre Dame — A series of computer simulations was conducted to determine the optimal method for reducing the chord length of large wind turbine blades while incorporating advanced flow control to offset the resulting loss in aerodynamic performance. The dominant building trend in the wind energy industry of turbines with progressively larger diameters provided the inspiration for this study. By reducing the chord along the inner region of the wind turbine blade, the total blade length could then be extended for the same mass of blade while limiting the additional costs and issues associated with increased blade length. In order to preserve certain geometric characteristics, the reduction in chord was achieved by scaling along the chord alone or by simply truncating the blade with a flat or circular cut. The aerodynamic requirements for the modified blade sections were to equal or better the total lift and the lift-to-drag ratio of the original blade sections. For this investigation, flow control consisted of plasma actuators located at a combination of the leading edge, maximum thickness, and trailing edge locations of the modified blade sections.

[1]Supported by General Electric Energy

9:18AM GD.00007 Noise reduction in a heated Mach 1.3 jet using plasma actuators[1] , MARTIN KEARNEY-FISCHER, MO SAMIMY, Ohio State University — Heating capabilities have recently been added to the free jet facility at the Gas Dynamics and Turbulence Laboratory (GDTL) of the Ohio State University using a storage-based off-line electric heater. This addition makes it possible to test the effectiveness of the localized arc filament plasma actuators (LAFPAs) for the purpose of either noise mitigation or mixing enhancement over a wide range of temperatures. These actuators have been used successfully at GDTL in high Reynolds number, high-speed unheated jets. The facility consists of an axisymmetric jet of exit diameter 2.54 cm with different nozzle blocks and variable jet temperature in an anechoic chamber. Previous work with a Mach 0.9 jet has shown significant increases in noise reduction effectiveness with increasing temperature. The next step is to determine if and how this trend continues in supersonic heated jets. A number of combinations of forcing azimuthal mode and temperature ratio at a wide range of forcing frequencies are experimented in a perfectly-expanded Mach 1.3 axisymmetric jet to examine LAFPAs effectiveness for far-field noise mitigation. The preliminary results to be presented indicate that the trends observed in the previous work continue in this supersonic jet.

[1]Supported by NASA.

9:31AM GD.00008 Low dimensional analysis of the flow over a three dimensional turret , MARLYN ANDINO, MARK GLAUSER, Syracuse University — The presence of turbulent flows in the path of a collimated beam produces degradation in its intensity hence abating its performance. A study of the flow physics around a cylindrical turret with the application of open-loop control has been performed. The evaluation of flow control performance is accomplished by analyzing the changes in the turbulent flow time/length scales across the turret surface for three cases. Even though these quantities are not direct measures of the aero-optics, literature suggest there is a strong relationship between them. Open-loop results demonstrate reductions in both RMS and Reynolds shear stress over the separated region. Results of the autocorrelations of the unsteady pressure sensors for the actuated low speed tests exhibit a more organized almost periodic behavior. We are interested in developing a low dimensional description of the flow field over a 3D turret through the use of velocity and unsteady surface pressure measurements. This will incorporate velocity/pressure correlations and mathematical tools such as Proper Orthogonal Decomposition and Modified Linear Stochastic Measurements to construct a low dimensional velocity-based closed-loop flow control model to estimate the flow states in real-time.

9:44AM GD.00009 Active Flow Control over a 3D Articulating Turret , RYAN D. WALLACE, PATRICK R. SHEA, Syracuse University, VAITHI THIRUNAVUKKARASU, HENRY CARLSON, Clear Science Corp, MARK N. GLAUSER, Syracuse University — An investigation of active flow control was conducted on an articulating 3D turret with a flat aperture using suction as the control input. Observability of the system was obtained by simultaneously sampled dynamic surface pressure at multiple locations around the aperture along with velocity flow field at the center plane of the turret. Both open loop and closed-loop control cases are examined for the purpose of reducing the turbulent fluctuations directly over the aperture. Open-loop control reduces the separation in the flow and decreases the levels of turbulence above the aperture. The large database of no control and open-loop control also provides a basis to develop closed-loop control. For closed-loop control, a simple proportional controller will feed back a low dimensional estimation of the flow based on dynamic surface pressure and velocity in an effort to improve upon the open-control cases.

9:57AM GD.00010 Active Flow Control Techniques for use on Three Dimensional Hemispherical Turrets , PATRICK SHEA, RYAN WALLACE, MARK GLAUSER, Syracuse University — Hemispherical turrets have been a topic of considerable interest over the past several decades with studies focusing on airborne optical device applications. Highly three dimensional, turbulent flows develop in the wake of a turret, especially when a flat, optical aperture is in place on the hemisphere. Both open and closed-loop flow control have been successfully applied to this geometry to control the turbulent flow over the aperture, but control of large scale structures in the wake using open-loop flow control have been less effective. Fluctuating loads on the turret, which can induce undesired structural loading, have been attributed to strong, turbulent fluctuations in the velocity of the turret wake. The current work involves developing a more robust active control system (both open and closed-loop using suction based actuators) that will not only allow for the control of the flow over the aperture as Syracuse University is currently studying, but will also allow for control of the large scale flow structures that develop in the wake of a turret.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GE Biofluids V: Cardiac Flows 101E

8:00AM GE.00001 Unsteady flows modeling using Smoothed Particle Hydrodynamics , SHAHROKH SHAHRIARI, IBRAHIM HASSAN, LYES KADEM, Mechanical and Industrial Engineering Department, Concordia University, Montreal, Canada — Cardiovascular diseases are the major cause of death in North America. Investigation of blood flow behavior in the cardiovascular system is, therefore, of great interest in biomedical engineering and cardiology. These kinds of flows are characterized by highly inertial pulsatile effects and deformable boundaries. The most important limitation of conventional numerical methods for simulating such flows is their main nature dependence on the process of mesh generation; distortion and remeshing that are numerically expensive. An alternative to overcome these limitations can be the new generation of numerical methods called meshfree methods. Smoothed Particle Hydrodynamics (SPH) is a Lagrangian meshfree method created originally to simulate astrophysical phenomena and later developed for applications in continuum solid and fluid mechanics. In this investigation, the potential of SPH method to model pulsating laminar flow in simplified (rigid) geometries found in the cardiovascular system such as left heart cavity and stenosed artery are examined. This work represents the first attempt to model internal pulsatile flows for a variety of Reynolds numbers using SPH. Although reaching physiological conditions still needs several improvements, SPH showed a good capability and could become a promising numerical method to simulate cardiovascular flows.

8:13AM GE.00002 Left ventricle of mammalian hearts optimzed for high hydrodynamic efficiency , LIANG GE, University of California San Francisco, ALI AZADANI, ELAINE TSENG — Mammalian hearts have four chambers: two atria (left and right) and two ventricles (left and right). The left ventricle (LV) is the primary pumping engine that pumps blood to all end organs of the body. The energetic efficiency of LV is therefore crucial for life. An important factor that contributes to the overall LV pumping efficiency is the hydrodynamic cost of blood flow within the LV chamber. LV blood flow is created by the cyclical expansion/contraction motion of LV wall and its hydrodynamic cost is certainly affected by the geometry and motion of LV wall. In this work we investigated the relationship between the hydrodynamic cost of LV filling/ejecting and LV geometry/motion and showed that the geometry and motion of mammalian hearts were optimized to minimize the hydrodynamic cost of LV blood flow.

8:26AM GE.00003 Early embryonic intra-cardiac flow fields at three idealized ventricular morphologies[1] , KEREM PEKKAN, MOHAMMAD JAMALY, BURAK KARA, Carnegie Mellon University, BRADLEY KELLER, University of Louisville, FOTIS SOTIROPOULOS, University of Minnesota — Pulsatile 3D multiple inlet/outlet flow within tiny (100-300μm dia) embryonic ventricles feature distinct intra-cardiac flow streams whose role in regulating the morphogenesis of spiral aorto-pulmonary septum has long been debated. The low Re number flow regimes limit mixing of these streams as replicated in our flow-visualization experiments with chick embryos. A state-of-the art high-resolution immersed boundary CFD solver which was developed for complex patient-specific cardiovascular internal flow problems is applied and optimized for this problem. Idealized tubular ventricles at 3 major embryonic stages (straight, C- and D- loops) are created by our sketch-based anatomical editing tool. CFD results are validated with PIV measurements acquired from a micro-fabricated C-loop stage replica and in vivo flow vis data from confocal microscopy. This model provided the inlet velocity profile for arterial models and flow fields at the inner curvature of embryonic hearts for different ventricular topologies are compared for off-design modes.

[1]Supported by American Heart Association.

8:39AM GE.00004 Simulations of heart mechanics over the cardiac cycle[1] , STAVROS TAVOULARIS, MATTHEW DOYLE, YVES BOURGAULT, University of Ottawa — This study is concerned with the numerical simulation of blood flow and myocardium motion with fluid-structure interaction of the left ventricle (LV) of a canine heart over the entire cardiac cycle. The LV geometry is modeled as a series of nested prolate ellipsoids and is capped with cylindrical tubes representing the inflow and outflow tracts. The myocardium is modeled as a multi-layered, slightly compressible, transversely isotropic, hyperelastic material, with each layer having different principal directions to approximate the fibrous structure. Blood is modeled as a slightly compressible Newtonian fluid. Blood flow into and out of the LV is driven by left atrial and aortic pressures applied at the distal ends of the inflow and outflow tracts, respectively, along with changes in the stresses in the myocardium caused by time-dependent changes in its material properties, which simulate the cyclic contraction and relaxation of the muscle fibers. Numerical solutions are obtained with the use of a finite element code. The computed temporal and spatial variations of pressure and velocity in the blood and stresses and strains in the myocardium will be discussed and compared to physiological data. The variation of the LV cavity volume over the cardiac cycle will also be discussed.

[1]Financial support was provided by NSERC and by OGSST.

8:52AM GE.00005 Fluid Structure Interaction simulation of heart prosthesis in patient-specific left-ventricle/aorta anatomies[1] , TRUNG LE, IMAN BORAZJANI, FOTIS SOTIROPOULOS, University of Minnesota, SAFL TEAM — In order to test and optimize heart valve prosthesis and enable virtual implantation of other biomedical devices it is essential to develop and validate high-resolution FSI-CFD codes for carrying out simulations in patient-specific geometries. We have developed a powerful numerical methodology for carrying out FSI simulations of cardiovascular flows based on the CURVIB approach (Borazjani, L. Ge, and F. Sotiropoulos, Journal of Computational physics, vol. 227, pp. 7587-7620 2008). We have extended our FSI method to overset grids to handle efficiently more complicated geometries e.g. simulating an MHV implanted in an anatomically realistic aorta and left-ventricle. A compliant, anatomic left-ventricle is modeled using prescribed motion in one domain. The mechanical heart valve is placed inside the second domain i.e. the body-fitted curvilinear mesh of the anatomic aorta. The simulations of an MHV with a left-ventricle model underscore the importance of inflow conditions and ventricular compliance for such simulations and demonstrate the potential of our method as a powerful tool for patient-specific simulations.

[1]Minnesota Supercomputing Institute, National Science Foundation and National Institutes of Health

9:05AM GE.00006 Volumetric velocity measurements on flows through heart valves , DANIEL TROOLIN[1], TSI Incorporated, DEVESH AMATYA[2], ELLEN LONGMIRE[3], University of Minnesota — Volumetric velocity fields inside two types of artificial heart valves were obtained experimentally through the use of volumetric 3-component velocimetry (V3V). Index matching was used to mitigate the effects of optical distortions due to interfaces between the fluid and curved walls. The steady flow downstream of a mechanical valve was measured and the results matched well with previously obtained 2D PIV results, such as those of Shipkowitz et al. (2002). Measurements upstream and downstream of a deformable silicone valve in a pulsatile flow were obtained and reveal significant three-dimensional features of the flow. Plots and movies will be shown, and a detailed discussion of the flow and various experimental considerations will be included. Reference: Shipkowitz, T, Ambrus J, Kurk J, Wickramasinghe K (2002) Evaluation technique for bileaflet mechanical valves. J. Heart Valve Disease. 11(2) pp. 275-282.

[1]Fluid Mechanics Division
[2]Biomedical Engineering
[3]Aerospace Engineering and Mechanics

9:18AM GE.00007 Fluid dynamics of aortic valve stenosis , ZAHRA KESHAVARZ-MOTAMED, Mechanical and Industrial Engineering Department, Concordia University, Montreal, Canada, NIMA MAFTOON, Mechanical and Industrial Engineering Department, McGill University, Montreal, Canada — Aortic valve stenosis, which causes considerable constriction of the flow passage, is one of the most frequent cardiovascular diseases and is the most common cause of the valvular replacements which take place for around 100,000 per year in North America. Furthermore, it is considered as the most frequent cardiac disease after arterial hypertension and coronary artery disease. The objective of this study is to develop an analytical model considering the coupling effect between fluid flow and elastic deformation with reasonable boundary conditions to describe the effect of AS on the left ventricle and the aorta. The pulsatile and Newtonian blood flow through aortic stenosis with vascular wall deformability is analyzed and its effects are discussed in terms of flow parameters such as velocity, resistance to flow, shear stress distribution and pressure loss. Meanwhile we developed analytical expressions to improve the comprehension of the transvalvular hemodynamics and the aortic stenosis hemodynamics which is of great interest because of one main reason. To medical scientists, an accurate knowledge of the mechanical properties of whole blood flow in the aorta can suggest a new diagnostic tool.

9:31AM GE.00008 Dynamic Energy Loss Characteristics in the Native Aortic Valve , CHOON HWAI YAP, Georgia Institute of Technology, LAKSMI P. DASI, Colorado State University, AJIT P. YOGANATHAN, Georgia Institute of technology — Aortic Valve (AV) stenosis if untreated leads to heart failure. From a mechanics standpoint, heart failure implies failure to generate sufficient mechanical power to overcome energy losses in the circulation. Thus energy efficiency-based measures are direct measures of AV disease severity, which unfortunately is not used in current clinical measures of stenosis severity. We present an analysis of the dynamic rate of energy dissipation through the AV from direct high temporal resolution measurements of flow and pressure drop across the AV in a pulsatile left heart setup. Porcine AV was used and measurements at various conditions were acquired: varying stroke volumes; heart rates; and stenosis levels. Energy dissipation waveform has a distinctive pattern of being skewed towards late systole, attributed to the explosive growth of flow instabilities from adverse pressure gradient. Increasing heart rate and stroke volume increases energy dissipation, but does not alter the normalized shape of the dissipation temporal profile. Stenosis increases energy dissipation and also alters the normalized shape of dissipation waveform with significantly more losses during late acceleration phase. Since stenosis produces a departure from the signature dissipation waveform shape, dynamic energy dissipation analysis can be extended into a clinical tool for AV evaluation.

9:44AM GE.00009 Fluid-structure interaction analysis of the flow through a stenotic aortic valve , HODA MALEKI, Mech. Eng. Dept., Concordia Univ., Montreal, Canada, MICHEL R. LABROSSE, Mech. Eng. Dept., Ottawa Univ., Ottawa, Canada, LOUIS-GILLES DURAND, Institut de Recherches Cliniques de Montréal, Montreal, Canada, LYES KADEM, Mech. Eng. Dept., Concordia Univ., Montreal, Canada — In Europe and North America, aortic stenosis (AS) is the most frequent valvular heart disease and cardiovascular disease after systemic hypertension and coronary artery disease. Understanding blood flow through an aortic stenosis and developing new accurate non-invasive diagnostic parameters is, therefore, of primarily importance. However, simulating such flows is highly challenging. In this study, we considered the interaction between blood flow and the valve leaflets and compared the results obtained in healthy valves with stenotic ones. One effective method to model the interaction between the fluid and the structure is to use Arbitrary Lagrangian-Eulerian (ALE) approach. Our two-dimensional model includes appropriate nonlinear and anisotropic materials. It is loaded during the systolic phase by applying pressure curves to the fluid domain at the inflow. For modeling the calcified stenotic valve, calcium will be added on the aortic side of valve leaflets. Such simulations allow us to determine the effective orifice area of the valve, one of the main parameters used clinically to evaluate the severity of an AS, and to correlate it with changes in the structure of the leaflets.

9:57AM GE.00010 A study of the pulsatile flow and its interaction with rectangular leaflets , RENE LEDESMA, ROBERTO ZENIT, GUILLERMO PULOS, Instituto de Investigaciones en Materiales, UNAM, Mexico — To avoid the complexity and limited understanding of the 3D pulsatile flow field through heart valves, a cardiac-like flow circuit and a test channel were designed to study the behavior of bidimensional leaflets made of hyperelastic materials. We study a simple 2D arrangement to understand the basic physics of the flow-leaflet interaction. Creating a periodic pressure gradient, measurements of leaflet deflection were obtained for different flow conditions, geometries and materials. Using PIV and Phase Locking techniques, we have obtained the leaflet motion and the time-dependent flow velocity fields. The results show that two dimensionless parameters determine the performance of a simple bi-dimensional valve, in accordance with the flow conditions applied: Π_1=f(sw)$^{1/2}$(E/ρ)$^{1/2}$ and Π_2=V/(2slw), where f is the pulsation frequency, V is the stroke volume, s, w and l are the dimensions on the leaftlet and E and ρ are the elastic modulus and density of the material, respectively. Furthermore, we have identified the conditions for which the fluid stresses can be minimized. With these results we propose a new set of parameters to improve the performance of prosthetic heart valves and, in consequence, to reduce blood damage.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GF Microfluidics: Electric Fields and Particles 101F

8:00AM GF.00001 Electro-hydrodynamic particle levitation on electrodes , EHUD YARIV, Technion — When colloidal particles deposit electro-phoretically onto a planar electrode, they slowly aggregate, eventually forming planar 2D crystalline structures. The attractive particle-particle interaction is due to electrokinetic flows associated with the particle Debye layer as well as the induced Debye layer surrounding the electrode. A common feature in the experimental observations is the small thickness of the particle-electrode gap separation, which was indeed reflected in the numerical figures employed hitherto in the existing numerical analyses. Here, we exploit it using singular perturbation methods. Thus, the fluid domain is separated into an "inner" gap region, where the electric field and flow strain rate are intensive, and an "outer" domain, consisting of the remaining fluid domain, where they are moderate. The inner region is analyzed using standard lubrication approximation, and the outer region is investigated using tangent-spheres coordinates. This method provides an analytic approximation for the hydrodynamic force that keeps the particle levitating against the action of gravity, as well as far-field approximations for the velocity decay, which agree with existing numerical simulations.

8:13AM GF.00002 Near-contact electrokinetic interactions between ideally polarizable particles , MOHAMMAD ABU HAMED, EHUD YARIV, Technion — When a zero-net-charge spherical particle is exposed to a uniformly applied electric field it polarizes, giving rise to an induced zeta potential distribution and a concomitant electro-osmotic flow field. Due to symmetry, the particle does not experience any electrophoretic motion. This symmetry is disturbed when an adjacent boundary (e.g. another particle or a channel wall) is introduced. This gives rise to boundary-driven particle motion, which is nonlinear in the applied field, approximately quadratic in it when it is weak. Using matched asymptotic expansions, we analyze electrokinetic interactions between a pair of ideally polarizable particles at small gap separations. When the field is applied perpendicular to their line of center, it tends to repel the particles away from each other. This repulsion is dominated by the pressure field within the gap, animated by the intense electric field there. The resulting pair interaction weakly diverges as an irrational power of the gap thickness. When the field is applied in parallel to the line of center, the electric field within the gap is exponentially small, and the pair interaction is bounded.

8:26AM GF.00003 DC electrokinetic transport of a cylindrical particle in a rectangular microchannel[1] , YE AI, Department of Aerospace Engineering, Old Dominion University, SANG W. JOO, School of Mechanical Engineering, Yeungnam University, ALI BESKOK, SHIZHI QIAN, Department of Aerospace Engineering, Old Dominion University — Electrokinetic transport of a cylindrical microparticle in a straight microchannel under direct current (DC) electric fields is numerically and experimentally investigated. DC dielectrophoresis (DEP) is taken into account in the proposed mathematical model, which is composed of the Navier-Stokes equations for the flow field and the Laplace equation for the electric field solved in an arbitrary Lagrangian-Eulerian (ALE) framework. Cylindrical particles experience an oscillatory motion under low electric fields. As the electric field increases, the induced DEP force acting on the particle gradually diminishes the oscillatory motion. Once the electric field is larger than a certain threshold value, the particle only translates with its axis parallel to the applied electric field after a short oscillatory motion. The numerical predictions are in good agreement with the experimental results.

[1]This work is supported, in part, by Ministry of Education, Science and Technology of Korea World Class University (WCU) Program (SWJ and SQ).

8:39AM GF.00004 Modeling Electrophoresis of Microtubules in Microchannels , XIAN LUO, Brown University, ALI BESKOK, Old Dominion University, GEORGE KARNIADAKIS, Brown University — We simulate the electrophoretic motion of individual microtubules in microchannels in order to obtain their anisotropic mobility and compare with recent experimental results (van den Heuvel et al., PNAS, 2007). We include for comparison simulation results for a circular cylinder with a similar "effective" radius, in order to examine how the surface roughness of microtubules affects the electrical double layer, the externally applied field, and hence the electrophoretic mobility. The simulation method is based on the smoothed profile method (SPM) – an immersive-boundary-like method –and spectral element discretization. The new method allows for arbitrary differences in the electrical conductivities between the charged surfaces and the ionized solution.

8:52AM GF.00005 Electrophoresis of deformable elastic particles , TONG GAO, T.N. SWAMINATHAN, HOWARD HU, University of Pennsylvania — Electrophoretic motion of a deformable dielectric elastic particle, having a fixed zeta potential,placed in an external electric field, has been numerically simulated. The potential field is solved in the fluid external to the particle, to compute the applicable Helmholtz-Smoluchowski slip boundary conditions on the particle surface. A constitutive equation is constructed for an incompressible neo-Hookean elastic solid where the extra stress tensor is assumed to be linearly proportional to the $Almansi$ strain tensor, to govern the deformation of the particle. A monolithic finite element solver which uses an Arbitrary Lagrangian-Eulerian moving mesh technique is then used to solve the velocity, pressure and stress field in both the fluid and solid phases simultaneously. The particle is initially elliptical and is aligned perpendicular to the direction of the applied electric field. Elastic deformation is observed as the particle moves. Two cases of zero and finite Reynolds number are examined to delineate the effect of the inertial terms on the deformation of the particle. The stress and pressure distributions on the particle surface are also compared with some analytical solutions.

9:05AM GF.00006 Effects of Ion Sterics and Hydrodynamic Slip on Electrophoresis of a Colloidal Particle , ADITYA KHAIR, TODD SQUIRES, Chemical Engineering, UC Santa Barbara — The classical theory of a spherical colloids' electrophoretic mobility is founded on the Poisson-Nernst-Planck (PNP) equations and assumes the standard hydrodynamic no-slip boundary condition at the fluid/solid interface. In the (common) limit of thin double-layers, the mobility has long been known to exhibit a maximum at some zeta potential, then decrease and asymptote to a constant value. Dukhin, O'Brien, White and others showed this to result from the importance of excess ionic surface conductivity within the double-layer. The fundamental assumptions that underpin this result are, however, subject to challenge: in recent years, a finite liquid/solid slip has been measured over a variety of surfaces, and the PNP equations predict physically impossible ion concentrations precisely at the high zeta potentials where the mobility maximum occurs. Here, we discuss the dramatic effect that hydrodynamic slip and finite-ion-size steric effects in double-layers have upon the electrophoretic mobility of spherical colloids, and therefore upon the interpretation of electrophoretic mobility measurements.

9:18AM GF.00007 Electrokinetic Traveling Waves in Non-Dilute Colloidal Dispersions[1] , CARLOS PEREZ, Mechanical Engineering, Arizona State University, JONATHAN POSNER, Mechanical Engineering, Chemical Engineering, Arizona State University — The existence of electrokinetically-driven, traveling waves in colloidal dispersions is presented. A non-dilute colloidal dispersion of 2 micron polystyrene microspheres are exposed to an ac electric field. Traveling waves consist of alternating regions of compressed and rarefied particle volume fraction that propagate through the dispersion parallel to the applied field. Colloids under the application of these ac fields have no net displacement, yet the travelling waves propagate at speeds at a tenth of the RMS electrophoretic velocity of individual particles. The collective dynamics of the colloids are described by the one dimensional, inviscid Burgers' equation. The waves originate from the modification of the colloid velocity due to the mobility's dependence on the local volume fraction and the particle electrokinetic polarization dipole interactions. The Burgers' equation analysis is used to predict the wave speed of the traveling waves.

[1]This work was sponsored by an NSF CAREER Award (CBET-0747917) with William Wendell Schultz as grant monitor.

9:31AM GF.00008 Aggregation and Coalescence of Emulsion Droplets via Electrohydrodynamic Flows , C.R. VIGO, W.D. RISTENPART, Univ. Calif. at Davis — Electrohydrodynamic (EHD) flows are known to cause rigid colloids to aggregate near electrodes [1]. Here we report that EHD flows also induce immiscible liquid droplets to aggregate and, for sufficiently strong electric fields, to coalesce. We measure the aggregation and coalescence rates of micron-scale olive oil droplets in water, and we interpret the coalescence rates in terms of a balance between EHD flow and repulsive colloidal scale (DLVO) forces. The results have broad implications for industrial processes in which trace amounts of immiscible oils need to be removed from aqueous solutions.

[1] Ristenpart, Aksay & Saville, J. Fluid Mechanics 575, 83, (2007).

9:44AM GF.00009 Electric field induced self assembly of floating rectangular plates , MUHAMMAD JANJUA, SAI NUDURUPATI, Lake Superior State University, IAN FISCHER, PUSHPENDRA SINGH, New Jersey Institute of Technology, NADINE AUBRY, Carnegie Mellon University — We show that an external electric field normal to a fluid-fluid interface can be used to self assemble rectangular plates floating on the interface and that the lattice spacing of the monolayer thus formed can be varied by changing the electric field intensity. In our experiments, a rectangular plate floats so that the contact line is pinned at the upper edge. Plates experience lateral forces due to capillarity which cause them to cluster. In the presence of an electric field, plates are also subjected to the repulsive electrostatic forces which, together with the attractive capillary forces, determine the equilibrium spacing of the monolayer. The interface profile around the plates is also modified by the electric field.

9:57AM GF.00010 Brownian Dynamics modeling of electrophoretic dsDNA-molecule separation using nanofluidic devices[1] , GHASSAN FAYAD, NICOLAS HADJICONSTANTINOU, MIT — We present a Brownian Dynamics model of electrophoretic separation of short (up to 7 persistence lengths) dsDNA molecules in nanofluidic devices. Our formulation uses the Worm-Like-Chain model with hydrodynamic interactions. Our simulation results are in good agreement with the experimental results of Fu et al. [*Phys. Rev. Lett.*, **97**, 018103, 2006] for realistic values of all physical parameters. We also find good agreement between our simulation results and the theoretical model of Li et al. [*Anal. Bioanal. Chem.*, **394**, 427, 2009] who proposed an asymmetric separation device that operates under the effect of an alternating electric field.

[1]This work was supported by the Singapore-MIT Alliance.

Monday, November 23, 2009 8:00AM - 9:57AM –
Session GG Microfluidics: Mixing 101G

8:00AM GG.00001 Formation of coherent structures in 3D laminar mixing flows , MICHEL SPEETJENS, Energy Technology Laboratory, Mechanical Engineering Department, Eindhoven University of Technology, P.O. Box 513, 5600 MB, Eindhoven, The Netherlands, HERMAN CLERCX, Fluid Dynamics Laboratory, Applied Physics Department, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands — Mixing under laminar flow conditions is key to a wide variety of industrial systems of size extending from microns to meters. Examples range from the traditional (and still very relevant) mixing of viscous fluids via compact processing equipment down to emerging micro-fluidics applications. Profound insight into laminar mixing mechanisms is imperative for further advancement of mixing technology (particularly for complex micro-fluidics systems) yet remains limited to date. The present study concentrates on a fundamental transport phenomenon of potential relevance to laminar mixing: the formation of coherent structures in the web of 3D fluid trajectories due to fluid inertia. Such coherent structures geometrically determine the transport properties of the flow and better understanding of their formation and characteristics may offer ways to control and manipulate the mixing properties of laminar flows. The formation of coherent structures and its impact upon 3D transport properties is demonstrated by way of examples.

8:13AM GG.00002 Streamlines and mixing patterns for drops in capillaries , FRANCOIS BLANCHETTE, University of California Merced — We present a theoretical and numerical investigation of streamlines and mixing patterns within drops flowing in capillaries. We study theoretically the limit case of purely viscous flow around a small drop, and find that recirculating regions are always present at the front and back of such drops. Using two-dimensional simulations, we visualize streamlines for larger drops, showing that the extent of these recirculating torii increases with drop size and decreases with Reynolds number. We study the mixing within drops as they are subjected to time-dependent shear, thus modeling a sinusoidal channel, and find that while cross-stream mixing is efficient, streamwise mixing is hindered by the front and back recirculating regions.

8:26AM GG.00003 Chaotic mixing in a plane channel with rotating arc walls , KAMAL EL OMARI, ADIL ACHHOUD, YVES LE GUER, University of Pau - France, LATEP TEAM — The effect of chaotic advection on the advection-diffusion of passive species is investigated for a new type of open flow mixer. This mixer is of active type with a perturbation of the flow imposed by three rotating circular arc walls (RAW) in a two-dimensional plane channel flow. Different steady flow topologies can be obtained depending on the respective directions of the RAW. Efficient stirring protocols were designed by the combination of some steady streamline patterns giving rise to chaotic mixing. The dynamical behavior of the mixing induced by these protocols were compared and discussed for different control parameters.

8:39AM GG.00004 The Ranz Stretch model and its extensions applied to mixing and reversibility , PAVITHRA SUNDARARAJAN, JOSEPH KIRTLAND, DONALD KOCH, ABRAHAM STROOCK, Cornell University — Mixing and separation are central to several chemical systems and are often carried out using microfluidics. Flow in a microfluidic device is usually in the laminar or Stokes regime. So mixing - a combination of stirring and diffusion - is often performed using chaotic flows which stir much faster than non chaotic flows. For separation of solutes of different diffusivities, Heller proposed the principle of Separation by Diffusive Irreversibility (SDI) which combines the reversibility of stirring and irreversibility of diffusion. Fundamental to both mixing and SDI is the interplay of convection and diffusion which is difficult to understand because of the challenging nature of convective coupling of solute concentration and the flow in the governing convective diffusion equation. Our approach is to use the Ranz model which observes the evolution of a single strand of concentration in the local linear flow. While this model captures the qualitative behavior of the chaotic and non chaotic flows, it fails to quantitatively predict the scaling of mixing and separation characteristics. Our goal is to identify a set of parameters that quantify the effects of stretch history and distribution and the presence of islands on mixing and separation. I will present a study of these effects, and the extensions of Ranz model incorporating these effects. I will compare the results from the model to the numerical simulation. The model will improve our understanding of mixing and irreversibility in Stokes flows.

8:52AM GG.00005 Long-term description of chaotic mixing induced by resonance phenomena , DMITRI VAINCHTEIN, SAHAND HARIRI AKBARI, Temple University, ROMAN GRIGORIEV, Georgia Institute of Technology — We present a quantitative long-term theory of resonant mixing in 3-D near-integrable flows. We illustrate that such resonance phenomena as resonance and separatrix crossings accelerate mixing by causing the jumps of adiabatic invariants. The resulting mixing can be described in terms of a single diffusion-type equation. We show what modifications must be made to accommodate the effects of the boundaries of the domain and possible correlations between the successive jumps.

9:05AM GG.00006 Development of an optimal mixer: a conceptual study , OLEG GUBANOV, LUCA CORTELEZZI, McGill University — We define as an optimal mixer a mixing device able to deliver a uniformly optimal mixing performance over a wide range of operating and initial conditions. We consider the conceptual problem of designing an optimal mixer starting from a reference mixing device, the sine flow. We show that the time-periodic sine flow performs poorly and erratically over most operating and initial conditions. In steps we modify the design of the reference mixer to obtain a mixing device whose performance, over the entire operating range, is as good as or better than the best performance of the sine flow. First, we optimize the time-sequence of the stirring velocity fields. The resulting mixer performs substantially better than the sine flow, but it is still suboptimal because the actuating system cannot control all the states. Second, we equip the sine flow with a new actuating system that allows optimized shifts of the stirring velocity fields in the cross-flow direction. This new actuating system is able to control all states. The resulting mixer delivers a suboptimal performance only at low operating conditions due to the use of a time-periodic stirring protocol. Finally, we obtain an optimal mixer by coupling the time and shift optimizations. We show that the resulting optimal mixer is able to deliver a nearly uniform optimal performance, insensitive to the geometry of the initial conditions, over the entire operating range.

9:18AM GG.00007 Effects of herringbone groove geometry on flow kinematics in a high aspect-ratio microchannel , VISHWANATH SOMASHEKAR, MICHAEL OLSEN, Iowa State University, MARK STREMLER, Virginia Tech — Passive mixing is often achieved in laminar microscale flows by driving fluid through microchannels with geometries that produce secondary flows and/or by splitting and recombining the fluid multiple times. One very successful microscale mixer design is the staggered herringbone mixer introduced by Stroock et al. (Science 2002) In the presented work, we consider very high aspect ratio (62:1) microchannels with a repeated staggered herringbone pattern spanning the entire width of the microchannel. Herringbone geometries with three different interior angles of the herringbone pattern (45, 90, and 135 degrees) were investigated. The flowfields within the herringbone mixers were determined using microscopic particle image velocimetry (microPIV). Velocity fields were measured at the midplane of the microchannel and at the groove-channel interface for Reynolds numbers based on microchannel hydraulic diameter of 0.08, 0.8, and 8. These wide microchannels produce secondary flow patterns that effectively split the fluid into parallel streams without having to fabricate physically separate channels.

9:31AM GG.00008 Breaking Regular Islands for Improved Mixing in an Electro-osmotic Device , RODOLPHE CHABREYRIE, Carnegie Mellon University, CRISTEL CHANDRE, Centre de Physique Theorique - CNRS, PUSHPENDRA SINGH, New Jersey Institute of Technology, NADINE AUBRY, Carnegie Mellon University, CARNEGIE MELLON UNIVERSITY TEAM, CENTRE DE PHYSIQUE THEORIQUE - CNRS TEAM, NEW JERSEY INSTITUTE OF TECHNOLOGY TEAM — Two-dimensional electro-osmotic flow with strong spatial and weak temporal variations of the zeta potential is investigated theoretically for the purpose of enhancing mixing in a microchannel. The flow is a superposition of a primary component and a perturbation. The primary flow, generated by the spatially periodic zeta potential, consists of recirculating rolls, while the perturbation arises due to a small time periodic variation of the zeta potential distribution. In this work, we propose a method that allows us to identify the values of the parameters which produce complete mixing. The method is based on tracking the linear stability of the main periodic orbits corresponding to the recirculating rolls of the primary flow. Poincaré maps, Lyapunov exponents and a box counting measure are computed to corroborate our results.

9:44AM GG.00009 Experimental study of mixing in low gravity by vibrations , ALIAKSANDR MIALDUN, YURII GAPONENKO, DENIS MELNIKOV, VALENTINA SHEVTSOVA, MRC, University of Brussels — In the absence of external forces, the diffusion process leads to the mixing of species on long time scale. The application of vibrations to a fluid system with density gradient causes relative flows inside the fluid. The aim of this study is to analyze the physical mechanism, by which vibrations affect the mixing process of two stratified miscible fluids. The rectangular cavity (10mm ×5 mm × 3mm) filled half-by-half with the two different miscible liquids is subjected to translational vibration. The direction of translational periodic vibrations with a constant frequency and amplitude is parallel to the interface between the two fluids. The system is kept at constant temperature. There is strong interplay between gravity and vibrational impact. To elucidate the vibrational mechanism the experiments were performed in parabolic flights organized by the European Space Agency. Parabolic flights provide repeated periods of approximately 20 seconds of reduced gravity preceded and followed by 20 seconds of hypergravity. The transient evolution of concentration field during microgravity time is investigated by optical digital interferometry. The analysis of the results shows that mixing and flow pattern in liquids depends not only on vibration stimuli but on the sharpness of the interface as well.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GH Drops IV: Breakup 101H

8:00AM GH.00001 Contraction of Asymmetric Newtonian Liquid Filaments , PATRICK MCGOUGH, KRISHNARAJ SAMBATH, SANTOSH APPATHURAI, PRADEEP BHAT, MICHAEL HARRIS, OSMAN BASARAN, Purdue University — Understanding the dynamics of satellite drops is important in several industrial applications involving drop formation including inkjet printing, electrospraying and atomization. The precursor to these satellite drops is a slender liquid filament that connects an about-to-form drop to the rest of the liquid in the nozzle. Once a filament is formed, it either contracts into a single satellite or breaks into multiple satellites, due to surface tension. Our understanding of the contraction of Newtonian filaments in a passive ambient fluid has improved greatly over the past two decades thanks to the numerical analyses of Schulkes (1996) and Notz and Basaran (2004) who modeled the filaments as cylinders that are terminated by two identical spherical caps. However, in many situations, the filament shapes at the onset of formation may not be symmetric as in the aforementioned studies. To improve our understanding of the fluid mechanics of contraction of such asymmetric filaments, we study here the recoil of filaments whose initial shapes are sections of tapered axisymmetric cones that are terminated by two unequal spherical caps. The dynamics are studied by both a 2-D analysis and a 1-D slender-jet analysis, and the results are summarized by constructing phase diagrams involving the dimensionless groups governing the dynamics.

8:13AM GH.00002 Effect of initial shape on contraction dynamics of Newtonian filaments , KRISHNARAJ SAMBATH, PATRICK MCGOUGH, SANTOSH APPATHURAI, PRADEEP BHAT, MICHAEL HARRIS, OSMAN BASARAN, Purdue University — Slender liquid filaments arise in a number of applications involving drop formation, atomization, and cloud physics. Under the action of surface tension, a filament either contracts into a single drop or breaks into multiple drops as it recoils. Our understanding of the contraction of Newtonian filaments in a passive ambient fluid has improved greatly over the past two decades thanks to the numerical analyses of Schulkes (1996) and Notz and Basaran (2004) who modeled the filaments as cylinders that are terminated by two identical hemispherical caps. However, in many situations, the initial shape of a filament may resemble more that of two unequal globular or spherical drops that are connected by a slender cylinder. The dynamics of contraction of such filaments are studied here by both a two-dimensional analysis and a one-dimensional slender-jet analysis, and the results are summarized by constructing phase diagrams involving the dimensionless groups governing the dynamics.

8:26AM GH.00003 Droplet formation from the breakup of micron-sized liquid jets , WIM VAN HOEVE, ARJAN VAN DER BOS, MICHEL VERSLUIS, JACCO SNOEIJER, Physics of Fluids, University of Twente, The Netherlands, MICHAEL P. BRENNER, School of Engineering and Applied Sciences, Harvard University, Cambridge, USA, DETLEF LOHSE, Physics of Fluids, University of Twente, The Netherlands, PHYSICS OF FLUIDS, UNIVERSITY OF TWENTE, THE NETHERLANDS TEAM, SCHOOL OF ENGINEERING AND APPLIED SCIENCES, HARVARD UNIVERSITY, CAMBRIDGE, USA TEAM — Droplet formation from the breakup of a liquid jet emerging from a micron-sized circular nozzle is investigated with ultra high-speed imaging at 1 million frames per second and within a lubrication approximation model [Eggers and Dupont, Phys. Rev. Lett. 262, 1994, 205-221]. The capillary time $\tau_c = \sqrt{\rho r^3/\gamma}$ is extremely small – of the order of 1μs. In the analyzed low Reynolds number regime the jet breakup is driven by surface tension forces only. Rayleigh breakup is not influenced by the surrounding air. The high- speed imaging results and those from the model calculation perfectly agree for various liquid viscosities and jet velocities, confirming a universal scaling law also for diminutive Rayleigh jets.

8:39AM GH.00004 Scaling in two-fluid pinch-off , CHRIS POMMER, RONALD SURYO, HARIPRASAD SUBRAMANI, MICHAEL HARRIS, OSMAN BASARAN, Purdue University, West Lafayette, IN 47909 — Two-fluid pinch-off is encountered when drops or bubbles of one fluid are ejected from a nozzle into another fluid or when a compound jet breaks. While the breakup of a drop in a passive environment and that of a passive bubble in a liquid are well understood, the physics of pinch-off when both the inner and outer fluids are dynamically active is inadequately understood. In this talk, the breakup of a compound jet whose core and shell are both incompressible Newtonian fluids is analyzed computationally by a method of lines ALE algorithm which uses finite elements with elliptic mesh generation for spatial discretization and adaptive finite differences for time integration. Pinch-off dynamics are investigated well beyond the limit of experiments set by the wavelength of visible light and that of various algorithms used in the literature. Simulations show that the minimum neck radius r initially scales with time τ before breakup as τ^α where α varies over a certain range. However, depending on the values of the governing dimensionless groups, this initial scaling regime may be transitory and, closer to pinch-off, the dynamics may transition to a final asymptotic regime for which $r \sim \tau^\beta$, where $\beta \neq \alpha$.

8:52AM GH.00005 Single drop fragmentation is the source of raindrops size distribution , EMMANUEL VILLERMAUX, BENJAMIN BOSSA, Aix-Marseille Universite, IRPHE — Like many natural objects, raindrops are distributed in size. By extension of what is known to occur inside the clouds, where small droplets grow by accretion of vapor and coalescence, raindrops in the falling rain at the ground level are believed to result from a complex mutual interaction with their neighbors. We show that the raindrops polydispersity, generically represented according to Marshall-Palmer's law, is quantitatively understood from the fragmentation products of non interacting, isolated drops. Both the shape of the drops size distribution, and its parameters are related from first principles to the dynamics of a single drop deforming as it falls in air, ultimately breaking into a dispersion of smaller fragments containing the whole spectrum of sizes observed in rain. The transformation is accomplished within a timescale much shorter than the typical collision time between the drops.

9:05AM GH.00006 Droplet breakup past an obstacle[1] , SUZIE PROTIERE, DAVE WEITZ, SEAS - Harvard University, HOWARD STONE, Princeton University — To investigate the transport of drops in a porous medium, we consider a model at the scale of an elementary event consisting of drop passing an obstacle in a microfluidic channel. We can thus observe the breakup process in a controlled way. We demonstrate that there exists an unstable situation for which a drop manages to pass the obstacle without breaking and define a critical value of the capillary number Ca* for which a drop will break. We also show that the obstruction dimensions play an important role in the breakup-non breakup transition. Finally we propose a model which describes the observed transition between breakup and non-breakup.

[1]Supported by Schlumberger-Doll Research, Cambridge MA.

9:18AM GH.00007 Production of ultra-small ink jet drops using drop-on-demand (DOD) drop formation , HAIJING GAO, Purdue University, QI XU, BP PLC, MICHAEL HARRIS, OSMAN BASARAN, Purdue University — The formation of drops having radii that are smaller than the radii of the nozzle from which they are ejected is an active area of research in drop-on-demand (DOD) ink jet printing. In the last decade, Chen and Basaran (Phys Fluids, 2002; US patent, 2003) showed experimentally and computationally that several fold reduction in drop radius R (an order of magnitude reduction in drop volume V) is possible by judicious use of waveform modulation in which one or more intrinsic time scales such as capillary time, time for vorticity diffusion, and time for piezo actuation are varied. In this paper, we report the results of a computational study through which we have uncovered a novel method for achieving a factor of 5-10 reduction in R (about two to three orders of magnitude reduction in V). Scaling arguments are also developed which yield a simple expression for the size of the ultra-small drops formed as a function of the governing dimensionless groups. Formation of such small drops using DOD technology may prove especially attractive in applications involving direct printing of flexible electronics and solar cells.

9:31AM GH.00008 Oscillations of a capillary switch used as a miniature opto-fluidic device , SANTHOSH RAMALINGAM, OSMAN BASARAN, Purdue University, West Lafayette — A capillary switch (CS) is a continuous volume of liquid consisting of a sessile and a pendant drop that are coupled through a liquid filled hole in a plate. When capillary force is much larger than body forces such as gravity, this simple, coupled interfacial system exhibits multiple equilibrium states beyond a critical volume. Owing to its extremely small size, and hence large curvature and highly spherical air-liquid interface, an oscillating CS can potentially be used as a variable focus liquid lens in MEMS devices. The dynamics of an oscillating CS are studied by solving the full 3D axi-symmetric or 2D Navier-Stokes equation using the Galerkin finite element method (G/FEM). Applying means of forcing such as oscillating the pressure in the gas surrounding the sessile (pendant) drop and vibrating the plate, modes of oscillation are identified from resonances observed during frequency sweeps. The shift in the frequencies of oscillation of lower modes due to changes in parameters such as liquid volume, plate thickness, and liquid viscosity and surface tension are also studied. Results are shown to agree well with experimental observations by Hirsa and coworkers.

9:44AM GH.00009 Effects of viscoelasticity on retraction of a sheared drop , SWARNAJAY MUKHERJEE, KAUSIK SARKAR, University of Delaware — The retraction of a sheared drop when either the drop or the matrix phase is Oldroyd B is investigated. The retraction is initially faster and later slower with increasing drop viscoelasticity. The initial faster relaxation of viscoelastic drops is due to inward pulling viscoelastic stresses at the drop tip and the later slowing down is due to the slowly relaxing viscoelastic stresses at the equator. The behavior is captured well by three model ODEs for two principal viscoelastic stresses (along the tip and equatorial directions) and the deformation. Matrix viscoelasticity slows the relaxation of a Newtonian drop right from the beginning because of the slow relaxation of stresses near the drop tip with increasing Deborah number. For drops sheared in supercritical conditions, when initially stretched beyond a certain length, relaxation leads to neck formation with two bulbous ends resulting in drop break-up, while for less stretching, it relaxes back to its spherical state.

9:57AM GH.00010 Dynamics of contracting viscoelastic filaments , MICHAEL HARRIS, SANTOSH APPATHURAI, PRADEEP BHAT, OSMAN BASARAN, Purdue University — Satellite drops are detrimental to many industrial applications involving the formation of viscoelastic drops including inkjet printing, DNA microarraying, and printing of flexible solar cells. The precursor to these satellite drops is a slender liquid filament that connects an about-to-form drop to the rest of the liquid in the nozzle. Once a filament is formed, it contracts due to surface tension. A filament may undergo further breakup during recoil. Whereas the contraction of Newtonian filaments in a passive ambient fluid is well understood (Schulkes 1996 and Notz and Basaran 2004), the contraction dynamics of viscoelastic filaments remains largely unexplored and is addressed in this presentation. Here the filament shape is idealized as an axisymmetric fluid cylinder terminated by hemispherical end-caps, and the conformation tensor formalism (Pasquali & Scriven 2002) is used to model the viscoelasticity. The dynamics of contracting filaments are then analyzed by means of both a well-benchmarked two-dimensional finite element algorithm (Notz et al. 2001, Chen et al. 2002) and a one-dimensional slender-jet algorithm (Padgett et al. 1996). Regions of the parameter space are identified where recoiling filaments give rise to either a single satellite drop or multiple satellites.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GJ Bubbles IV: Microbubbles, Bubble Motion 101I

8:00AM GJ.00001 Microbubbles transfer and segregation mechanisms in turbulent upward/downward channel flow. , DAFNE MOLIN, ANDREA GIUSTI, ALFREDO SOLDATI, Dept. of Energetics and Flow Machinery, University of Udine, Udine (Italy) — The dispersion of microbubbles in a turbulent channel flow is studied by means of direct numerical simulation (DNS), both in upward and downward flow with a two-way coupling approach. Microbubbles dispersion shows a sharply distinct behavior in the two flow cases: in upward flow bubbles tend to accumulate and segregate near the walls, whereas in the downward flow they tend to segregate in the center of the channel. This different spatial distribution, which is due to the interplay between turbulent wall transfer mechanisms and the local fluid forces acting on bubbles (especially the lift force), is expected to have an influence on the flow field. In this work, we present detailed results from a systematic analysis on the effect of the different forces acting on bubbles and how the flow statistics are modified by the presence of bubbles.

8:13AM GJ.00002 Study of dynamics of microbubble generation in microchannels , RYOJI MIYAZAKI, YOSHIMASA GOSHIMA, SHU TAKAGI, YOICHIRO MATSUMOTO, The University of Tokyo — The novel technique to generate micrometer-order bubbles was developed by using a microchannel with a squeezed T-junction, and the mechanism of bubble generation was investigated by using a high-speed camera with 106 Hz and the microscopy. The experiments were conducted by using three kinds of channels with the different cross-section size, and pure water, ethanol and silicon oil were selected as the liquid phase to examine the effect of the cross-section size of the channels and the physicality of the liquid phase. The liquid velocity at the T-junction and the gas pressure were set at 0.1$\sim$3.0 m/s and 10$\sim$200 kPa, respectively. The experimental results indicate that the proposed technique realizes to generate 10$\sim$30 μm diameter bubbles, and the diameter of the generated bubble becomes smaller with an increase of the liquid velocity, until limit points of bubble generation. From the experiment near the bubble generated limit, liquid pressure balances with the gas pressure and the Laplace pressure under the bubble generated limit, and the bubble diameter is dominated by Weber number which is defined using an equivalent diameter of the cross-section of the channel and the mean velocity of the liquid phase.

8:26AM GJ.00003 Numerical study on the behavior of a microbubble encapsulated by hyperelastic membrane in the ultrasound field , YUNQIAO LIU, Department of Mechanical Engineering, The University of Tokyo, KAZUYASU KAZUYASU SUGIYAMA, SHU TAKAGI, YOICHIRO MATSUMOTO — The surface stability problem of a microbubble encapsulated by a neo-Hookean hyperelastic membrane is numerically addressed. To predict this nonlinear behavior, the continuity equation and the Navier-Stokes equation are directly solved by means of the boundary-fitted finite-volume method on an orthogonal curvilinear coordinate system. The force balances of the membrane are derived from the traction jump condition, coupling with the in-plane tensions and transverse shear tension. The bubble is insonified by an ultrasound pulse at frequency of 1MHz and consisting of a burst of 10 cycles. The strain-softening features are presented referring to a linear model based on the Rayleigh-Plesset equation. For small acoustic amplitude, the result based on the neo-Hookean model is in good agreement with that on the linear model. With the increasing of oscillatory amplitude, the neo-Hookean membrane bubble shows an enhanced strain-softening effect – larger expansion, smaller contraction and higher harmonics during contraction. In addition, the neo-Hookean membrane bubble presents second-order shape instability. At the same time, this second-order mode shows subharmonics characteristics, which is considered as a potential medical application for ultrasonic imaging.

8:39AM GJ.00004 Effect of Ultrasound-Induced Bjerknes Force on the Dynamics of Microbubbles. Interaction with Saffman's lift[1] , ALBERTO ALISEDA, CHERYN ENGEBRECHT, Department of Mechanical Engineering. University of Washington — We will discuss results of experiments on the trajectories of Ultrasound Contrast Agents immersed in low Reynolds steady flow in a pipe. The microbubbles are subject to hydrodynamic forces, and are under the effect of external ultrasound forcing propagating normal to the flow direction. High speed visualization of the microbubbles trajectories shows significant deviations in the direction perpendicular to the flow. This displacement is due to the balance of the Bjerknes force and Saffman's lift. The dependency of the value and orientation of the microbubbles trajectories indicates a rich mechanism for the coupling between these two forces. In the absence of ultrasound excitation, Saffman's lift forces the microbubbles towards the wall. The volume oscillations induced on the microbubble by the propagating ultrasonic pressure waves significantly modify the lift, reversing its direction and making it away from the wall.

[1]Supported by NSF grant CBET-0748133.

8:52AM GJ.00005 Numerical Simulation of a Bubble Bouncing with a Free Surface , TOSHIYUKI OYAMA, SHINTARO TAKEUCHI TEAM, SHU TAKAGI TEAM, YOICHIRO MATSUMOTO TEAM — The paper presents a numerical study of a bubble-bouncing with a free surface using a three dimensional front-tracking method. According to our preliminary study, the bubble-free surface interaction is summarized as follows. The bubble becomes slightly oblate as it propels upward, and the bubble starts contacting at the side, rather than the top, to the elevated free surface. Then the liquid in film between the bubble and free surface is gradually drained until the bubble reaches the highest position. Finally, the bubble bounces back from the free surface due to the stored energy on the both of the surfaces and the self-induced flow field. We focus in the rebound depth, and duration time of bubble-free surface contact (contact time, hereafter). The contact time measured from the distance between the bubble center and free surface exhibits -0.5 power of surface tension coefficient, whereas the contact time based on the distance between the bubble top and free surface was found to be insensitive to surface tension coefficient. In the presentation, we also discuss the velocity field within the liquid film and the time-dependency of the film volume.

9:05AM GJ.00006 Multiscale interactions of bubbles with free vortex flows[1] , JUSTIN FINN, EHSAN SHAMS, SOURABH APTE, Oregon State University — We simulate bubble and particle interactions with several types of free vortex flows using both a Discrete Element Model (DEM) and a fully resolved approach. In the DEM approach, DNS is used with Lagrangian particle tracking to compute the motion of a subgrid scale dispersed phase. The *volumetric* displacement of the fluid by the dispersed phase is modeled along with interphase momentum-exchange for more realistic coupling of the dispersed phase to the flow. In the fully resolved approach, a fictitious domain technique is used with refined grids to directly compute the motion of the dispersed phase to obtain high fidelity solutions. First, both approaches are used to simulate bubble entrainment into a stationary Gaussian vortex [Oweis et al. 2005]. Next, bubble entrainment and interaction with a traveling vortex tube [Sridhar & Katz 1999] is simulated using the DEM approach. Finally, a viscous falling 'blob' of particles is simulated [Walther & Koumoutsakos 2001, Mitts 1995], where the dispersed phase generates and interacts with a 3D vortex ring. The results show that the less expensive DEM approach with volumetric coupling is able to capture clustering induced flow distortion, while the fully resolved approach gives insight into dispersed phase scale interactions with the flow.

[1]This work was partially supported by ONR: N000140610697.

9:18AM GJ.00007 Power spectral density in mono-dispersed bubbly flows , SANTOS MENDEZ, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — An experimental study was carried out to determinate the power spectral density functions of mono-dispersed bubbly flows in a vertical channel using flying hot-film anemometry. To improve the phase discrimination technique, an optic fiber was attached to the hot-film sensor. In this manner, it was possible to clearly separate the erroneous signals caused by bubble collision with the sensor. A special array of capillaries was used to produce nearly mono-dispersed flow. Measurements were performed with gas fractions up to 6%. The power spectral density distributions were found to have a good qualitative agreement with those obtained by other authors. Depending on the values of the Reynolds number and gas volume fraction a progressive change from a $-5/3$ to -3 decay was observed.

9:31AM GJ.00008 Self propulsion of bubbles in wedge-shaped geometries , HOWARD STONE, Princeton University, THIBAULT SCOARNEC, ANN LAI, MATHILDE REYSSAT, Harvard University — Self propulsion of bubbles and drops can be created by geometrically forcing capillary pressure gradients. We investigated such self propulsion experimentally by confining long bubbles in flat wedge-shaped geometries that have rectangular cross sections and are closed at both ends. The bubble moves from the narrow end toward the wider end with a speed that monotonically decreases in time. The fluid motion past the bubble occurs through the corners between the bubble and walls of the rectangular cross-section, so that the fluid flow is fully three dimensional. In order to quantitatively describe the motion of the bubble we introduce a one-dimensional model in the spirit of lubrication theory. The predictions of the model are in very good agreement with the experimental measurements and capture the variations with bubble size, wedge angle, and viscosity of the continuous phase.

9:44AM GJ.00009 Change of *Re* dependency of single bubble 3D motion by surface slip condition in surfactant solution[1] , YOSHIYUKI TAGAWA, AMI FUNAKUBO, SHU TAKAGI, YOICHIRO MATSUMOTO, The Univ. of Tokyo, THE UNIV. OF TOKYO TEAM — Path instability of single bubble in water is sensitive to surfactant. One of the key effects of surfactant is to decrease bubble rising velocity (i.e. increase drag) and change bubble slip condition from free-slip to no-slip. This phenomenon is described as Marangoni effect. However, the surfactant effect to path instability is not fully investigated. In this research, we measured bubble 3D trajectories and velocity in dilute surfactant solution to reveal the relation between 3D motion mode and slip condition. Experimental parameters are types of surfactants, concentrations and bubble sizes. Bubble motions categorized as straight, spiral or zigzag are plotted on two-dimensional field of bubble Reynolds number *Re* and normalized drag coefficient C_D^* which is strongly related to surface slip condition. Range of *Re* is from 200 to 1000 and C_D^* is from 0 to 1. Our results show that when C_D^* equals 0 or 1 (free-slip condition or no-slip condition, respectively), bubble motion mode is changed by *Re*. However when C_D^* is 0.5, bubble motion is always spiral. It means that *Re* dependency of bubble motions is strongly affected by slip condition. We will discuss its mechanism in detail in our presentation.

[1]Grant-in-Aid for JSPS Fellows

9:57AM GJ.00010 Slowing down bubbles with sound , CEDRIC POULAIN, Commissariat à l'Energie Atomique, France, REMIE DANGLA, Ecole Polytechnique, France, MARION GUINARD, Ecole Centrale Paris, France — We present experimental evidence that a bubble moving in a fluid in which a well-chosen acoustic noise is superimposed can be significantly slowed down even for moderate acoustic pressure. Through mean velocity measurements, we show that a condition for this effect to occur is for the acoustic noise spectrum to match or overlap the bubble's fundamental resonant mode. We render the bubble's oscillations and translational movements using high speed video. We show that radial oscillations (Rayleigh-Plesset type) have no effect on the mean velocity, while above a critical pressure, a parametric type instability (Faraday waves) is triggered and gives rise to nonlinear surface oscillations. We evidence that these surface waves are subharmonic and responsible for the bubble's drag increase. When the acoustic intensity is increased, Faraday modes interact and the strongly nonlinear oscillations behave randomly, leading to a random behavior of the bubble's trajectory and consequently to a higher slow down. Our observations may suggest new strategies for bubbly flow control, or two-phase microfluidic devices. It might also be applicable to other elastic objects, such as globules, cells or vesicles, for medical applications such as elasticity-based sorting.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GK Multiphase Flows III 101J

8:00AM GK.00001 Multiscale modeling of non-homogenous flows with non-Newtonian properties , ARTURO FERNANDEZ, Worcester Polytechnic Institute — A new multiscale approach to modeling non-homogenous flows where non-Newtonian effects are significant will be discussed. The computations are carried out by combining an immersed-boundary-method, able to capture the response of non-homogenous systems, with Brownian Dynamics able to predict the local properties. The exchange of information between the continuum-based and Brownian Dynamics models, which capture the system properties at different scales, is done through the velocity gradient and stress state tensors. The stress-state tensor estimated with the Brownian Dynamics simulations is introduced in the solution of the front-tracking method, whereas the velocity gradient state is used to estimate the local properties. How to achieve an adequate computational cost will also be discussed. The methodology is validated for two different problems: (i) the deformation of a Newtonian drop immersed in a simple shear flow and suspended in a viscoelastic fluid; (ii) the buoyancy of air bubbles in a viscoelastic fluid.

8:13AM GK.00002 Single-equation versus multi-equation models in simulation of material flows , XIA MA, DUAN ZHANG, PAUL GIGUERE, QISU ZOU, Los Alamos National Laborary — When considering interactions of two pieces of different materials, often a single momentum equation is used; and different materials are treated as two different species of a solid material. The stress in the momentum equation is calculated differently depending on the material occupying the point. This approach is limited when considering breakup of the materials into pieces with typical size smaller than numerical grid resolution. After the breakup, one would prefer to use a two-equation model to simulate the flow of the debris of the two solid materials. It is a significant issue when and how to switch from single-equation mode to the two-equation model. A different approach is to start with a two-equation model, and to treat the system as continuous two-phase system before the material breakup. When material breakup happens, the equation system has a smooth transition into disperse two-phase flows. The issue is then, how this two-equation approach compared with the single equation approach before the material breakup. What material interaction model is needed for such numerical calculation? The present paper tries to answer some of these questions using numerical examples.

8:26AM GK.00003 Multiscale Issues in DNS of Multiphase Flows[1] , GRETAR TRYGGVASON, SIJU THOMAS, JIACAI LU, BAHMAN ABOULHASANZADEH, Worcester Polytechnic Institute — In spite of the enormous information and understanding that DNS are providing for relatively complex multiphase flows, real systems provide challenges that still limit the range of situations that can be simulated, even when we limit our studies to systems well described by continuum theories. The problem is, as one might expect, one of scale. Starting with simulations where the "dominant small-scales" are fully resolved, it is frequently found that multiphase flows also can generate features much smaller than the dominant flow scales, consisting of very thin films, filaments, and drops. Frequently there is a clear separation of scales between these "features," usually inertia effects are relatively small for the local evolution, and in isolation these features are often well described by analytical models. Here we describe the use of a thin film model to account for unresolved features of the flow. By using a semi-analytical model for the flow in the film beneath a drop sliding down a sloping wall, we capture the evolution of films that are too thin to be accurately resolved using a relatively coarse grid that is sufficient to resolve the rest of the flow. Extensions of these ideas to flows with mass and heat transfer as well as phase change and chemical reactions are also discussed.

[1]Research supported by NSF and Sandia National Laboratory.

8:39AM GK.00004 Smoothed particle hydrodynamics applied to multiphase flows , MARION VANCE, KYLE SQUIRES, Arizona State University — Fully Lagrangian numerical simulations of multiphase flows are performed using a numerical approach that is a variation of smoothed particle hydrodynamics. The momentum conservation equation for the constant mass fluid elements is described using the Boltzmann transport equation and particle phase space probability density function. Analogous to the familiar forms of continuum fluid mechanics, the acceleration of a fluid element is due to the gradient of local kinetic stress, the constitutive terms of which are determined following expansion methods from kinetic theory of a dense gas. Including first-order terms, the acceleration of fluid elements is proportional to local particle density and relative velocity, and those elementary forces tend to drive the system towards an equilibrium state. The fundamental restoring and dissipative particle forces are shown to model familiar pressure, viscous, and surface tension effects at the macroscopic scale. The method is applied to test problems that include the wall- bounded flow of two nearly immiscible fluids, the rise of bubbles in an infinite quiescent liquid, and hard sphere sediment transport. Simulations are performed in both two and three dimensions, and the observations are compared to published results.

8:52AM GK.00005 A Comparison of Multiphase LBGK and MRT LBE Models , YAN PENG, LI-SHI LUO, Old Dominion University — One undesirable feature of LBE methods as diffuse interface methods is the existence of parasitic currents. Recently, Lee and Fischer have shown that if the potential form of the intermolecular force is used, the parasitic currents can be eliminated. In their study, the LBGK collision model is used. As we know that multiple-relaxation-time (MRT) collision model has a number of advantages over the lattice Bhatnagar-Gross-Krook (LBGK) model. In this study, we will replace the LBGK with MRT collision model. We compared the stability and Galilean invariance of the two models. The test case is a circular bubble. We found that LBGK is very sensitive to the initial given density values. For the Galilean invariance property, we first get the converged equilibrium solution. Then we add an external velocity. We found that LBGK scheme diverges even a very small velocity is given. From these comparisons, we conclude that MRT is more stable and preserve Galilean invariance better than LBGK.

9:05AM GK.00006 ABSTRACT WITHDRAWN –

9:18AM GK.00007 Effects of confinement on a rotating sphere[1] , QIANLONG LIU, ANDREA PROSPERETTI, Johns Hopkins University — The hydrodynamic force and couple acting on a rotating sphere in a quiescent fluid are modified by nearby boundaries with possible consequences on spin-up and spin-down times of particles uspended in a fluid, their wall deposition, entraiment and others. Up to now, the vast majority of papers dealing with these problems have considered the low-Reynolds-number regime. This paper focuses on the effect of inertia on the hydrodynamic interaction of a spinning sphere with nearby boundaries. Rotation axes parallel and perpendicular to a plane boundary as well as other situations are studied. Several steady and transient numerical results are presented and interptreted in terms of physical scaling arguments. The Navier-Stokes equations for an incompressible, constant-property Newtonian fluid are solved by the finite-difference PHYSALIS method. Among the noteworthy features of this method are the fact that the no-slip condition at the particle surface is satisfied exactly and that the force and torque on the sphere are obtained directly as a by-product of the computation. This feature avoids the need to integrate the stress over the particle surface, which with other methods is a step prone to numerical inaccuracies. A locally refined mesh surrounding the particle is used to enhance the resolution of boundary layers maintaining a manageable overall computational cost.

[1]This research is supported by NSF under grants CBET-0625138 and CBET-0754344.

9:31AM GK.00008 A linear spatial stability analysis of liquid-gas rotating co-flowing jet , YAOHONG WANG, MARK SUSSMAN, M.Y. HUSSAINI, Department of Mathematics, Florida State University — We present a linear spatial stability analysis of a liquid-gas rotating co-flowing jet. The parallel mean velocity is computed as a function of the radial coordinate by solving the coupled liquid-gas Navier-Stokes equations in a cylindrical coordinate system. A multi-domain Chebyshev spectral collocation method is applied to the perturbed Navier-Stokes equations (linearized about the mean parallel flow). Both axisymmetric and helical modes are considered. Numerical calculations are performed to obtain the growth rates and frequencies of the most unstable modes. The effect of density ratio, viscosity ratio and surface tension are discussed.

9:44AM GK.00009 Two Types of Equations for Nonlinear Wave Propagation in a Liquid Containing Microbubbles , TETSUYA KANAGAWA, Division of Mechanical and Space Engineering, Hokkaido University, TAKERU YANO, Department of Mechanical Engineering, Osaka University, MASAO WATANABE, SHIGEO FUJIKAWA, Division of Mechanical and Space Engineering, Hokkaido University — Weakly nonlinear propagation of one-dimensional dispersive waves in mixtures of a liquid and a number of spherical gas bubbles are theoretically investigated based on two-fluid averaged equations derived by the present authors. A set of equations consists of the conservation laws of mass and momentum for gas and liquid phases, and Keller's bubble dynamics equation. The compressibility of liquid leads to the wave attenuation due to bubble oscillations. By using the appropriate scaling of physical parameters and the method of multiple scales, two types of equations for nonlinear wave propagation in long ranges are derived. In a moderately low frequency band, the behavior of weakly nonlinear waves is governed by the KdV-Burgers equation. On the other hand, in a moderately high frequency band, the nonlinear modulation of quasi-monochromatic wave train is governed by the nonlinear Schroedinger equation with an attenuation term.

9:57AM GK.00010 Fluctuations in number and volume fraction in granular and multiphase flows: implications for theory and modeling , SHANKAR SUBRAMANIAM, Department of Mechanical Engineering, Iowa State University — Fluctuations in the number of particles, and consequently the fraction of volume occupied by them, are observed in experiments as well as simulations of granular and multiphase flows. The mathematical representation of these fluctuations is described, and compared with the standard average number density representation in kinetic theory of granular and gas-solid flow. Implications for the strong and weak forms of the conservation laws of hydrodynamic quantities are discussed, and this leads to possible approaches to model the effect of fluctuations. The manifestation of fluctuations in current closure models is examined using data from direct numerical simulation. Implications for the stability analysis of gas-solid flows, and the stability limits calculated from reduced statistical representations are discussed.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GL Particle Laden Flows II: Turbulence 200A

8:00AM GL.00001 Hybrid RANS/LES of particle-laden turbulent flows , MARCEL ILIE, University of Central Florida, STEFAN LLEWELLYN SMITH, University of California, San Diego — One of the main issues in using large-eddy simulation (LES) for high Reynolds number flows in bounded domains is the requirement of very fine grid resolution near walls. We present a hybrid RANS/LES method in conjuction with a Lagrangian particle tracking algorithm, for the numerical prediction of particle-laden turbulent flows. The hybrid RANS/LES methodology aims to reduce the high computational effort of wall-resolved LES. This approach is based on the concept of dividing the simulation into a near-wall RANS part and an outer LES part, and allows the thickness of the near-wall RANS layer to be chosen freely. The near-wall layer is interfaced to the outer LES region using compatibility conditions for velocity and turbulent viscosity across the interface that are extracted dynamically as the simulation progresses. The influence of parameters such as particle shape and size, particle density and flow Reynolds number on the particle dispersion and total deposition is examined. Particles of fiber shape are more prone to deposition. Total particle deposition increases with the particle size, density and Reynolds number.

8:13AM GL.00002 Simulation of turbulent flow laden with finite-size spherical particles , HUI GAO, LIAN-PING WANG, University of Delaware — Particle-laden turbulent flow is of importance to many engineering applications and natural phenomena. Most previous studies utilize the point particle approach to study the effects of particles on the carrier turbulence, under the assumptions that the particle size is significantly smaller than the smallest turbulence length scale and the particle volume fraction is low. The present study focuses on the motion and hydrodynamic interactions of finite-size freely moving particles in a turbulent background flow. A mesoscopic lattice Boltzmann approach is applied to simulate a homogeneous isotropic turbulence and to realize the no-slip boundary condition on the boundary of each moving particle. The short-range lubrication force not resolved by the simulation is represented by a model in terms of particle relative location and velocity. The change of energy spectrum compared with the particle-free turbulence is discussed, as well as the time evolution of the turbulent kinetic energy and the dissipation rate. The effects of varying particle size, volume fraction, and particle-to-fluid density ratio will also be examined.

8:26AM GL.00003 High-inertia particle acceleration statistics in a turbulent channel flow , VALENTINA LAVEZZO, ALFREDO SOLDATI, Dept. of Energetics and Flow Machinery, University of Udine, Udine, Italy, ZELLMAN WARHAFT, LANCE COLLINS, Sibley School of Mechanical and Aerospace Engineering, Cornell University, Ithaca, NY — Recent experiments in a turbulent boundary layer (Gerashchenko et al., 2008) have shown that the variance in the acceleration fluctuations of small, heavy particles in the near wall region increases with increasing inertia, contrary to the trend found for homogeneous and isotropic turbulence. In a previous study, we ran direct numerical simulations (DNS) of inertial particles in a channel flow to show how this phenomenon is related to the coupling of particle motion with shear and gravity. In this work, we extended the DNS to a much broader range of particle Stokes number (20, 40 and 100). The trend for the mean and variance of the particle-acceleration statistics at these much higher Stokes numbers are consistent with what previously was found for homogeneous and isotropic turbulence. We attribute this behavior to the inertial filtering by the particles of the underlying turbulent flow, as though at these higher Stokes numbers particles sample a more nearly isotropic flow field. The effect of gravity also has been considered and will be presented in detail.

8:39AM GL.00004 Direct Numerical Simulation of dense particle-laden turbulent flows using immersed boundaries , FAN WANG, OLIVIER DESJARDINS, Univ. of Colorado at Boulder — Dense particle-laden turbulent flows play an important role in many engineering applications, ranging from pharmaceutical coating and chemical synthesis to fluidized bed reactors. Because of the complexity of the physics involved in these flows, current computational models for gas-particle processes, such as drag and heat transfer, rely on empirical correlations and have been shown to lack accuracy. In this work, direct numerical simulations (DNS) of dense particle-laden flows are conducted, using immersed boundaries (IB) to resolve the flow around each particle. First, the accuracy of the proposed approach is tested on a range of 2D and 3D flows at various Reynolds numbers, and resolution requirements are discussed. Then, various particle arrangements and number densities are simulated, the impact on particle wake interaction is assessed, and existing drag models are evaluated in the case of fixed particles. In addition, the impact of the particles on turbulence dissipation is investigated. Finally, a strategy for handling moving and colliding particles is discussed.

8:52AM GL.00005 Experimental Investigation of Charged Inertial Particles in Turbulence[1] , HANSEN NORDSIEK, JIANG LU, EWE WEI SAW[2], RAYMOND SHAW, Department of Physics, Michigan Technological University — We report results from experiments aimed at studying the interactions of electrically charged inertial particles in homogeneous, isotropic turbulence. Conditions are selected to investigate the effects of mutual electrostatic repulsion of particles on their dynamics. We measure droplet clustering and relative velocities. The experiments are carried out in a laboratory chamber with nearly homogeneous, isotropic turbulence. The turbulence is characterized using LDV and 2-frame holographic particle tracking velocimetry. We seed the flow with charged particles and use digital holography to obtain 3D particle positions and velocities. From particle positions, we investigate the impact of mutual electrostatic repulsion on inertial clustering through the calculation of the radial distribution function (RDF). Specifically, repulsion overcomes inertial clustering below a shielding length as seen by a strong reduction in the RDF.

[1]Research supported by the US National Science Foundation.

[2]Now at Max Planck Institute for Dynamics and Self Organization.

9:05AM GL.00006 Effect of filtering on inertial particle clustering in homogeneous isotropic turbulence , BAIDURJA RAY, LANCE COLLINS, Cornell University — The use of large-eddy simulation (LES) to represent inertial particles in a turbulent flow field requires a model for the effect of the subfilter eddies on the particle motion. A particularly challenging aspect of this modeling is correctly capturing particle clustering, which is driven principally by the small-scale eddies that have been filtered in a LES. In this paper, we investigate this problem by performing direct numerical simulations of homogeneous isotropic turbulence with inertial particles and compare the results to particles moving through a low-pass filtered velocity field. The filtering is done in wavenumber space and is akin to a 'perfect' LES in that there is no subgrid model. We look at the two-particle radial distribution function (RDF) and the relative velocity probability density function (PDF) at different separation distances. We find that both the RDF and relative velocity PDF change substantially in response to the filtering. In particular, the level of clustering can be suppressed or enhanced depending on the value of the Stokes number. The spatial scales of the clustering are also affected. The results suggest requirements that a subfilter model should satisfy to correctly reproduce the RDF and relative velocity PDF. Such information will assist the future development of a LES model for inertial particles.

9:18AM GL.00007 On the effects of Taylor-lengthscale size particles on isotropic turbulence , F. LUCCI, University of California, Irvine, A. FERRANTE, University of Washington, S. ELGHOBASHI, University of California, Irvine — The effects of spherical particles of Taylor-lengthscale size ($d \sim \lambda$) on isotropic turbulence are studied via DNS. A mesh of 256^3 grid points is used with an initial microscale Reynolds number $Re_{\lambda 0} = 75$. The flow around 6400 freely-moving particles is fully resolved using the Immersed Boundary method. The maximum volume fraction of the particles is $\phi_v = 0.1$. The maximum density ratio is $\rho_p/\rho_f = 10$ which corresponds to a mass fraction $\phi_m = 1$. Our results show that particles with diameter $d \sim \lambda$ always reduce the turbulence kinetic energy (TKE), mostly by enhancing its dissipation rate, $\varepsilon(t)$. The augmented dissipation rate exceeds $\Psi_p(t)$, the rate of increase of TKE due to the two-way coupling force imparted by the particles on the surrounding fluid. The increased dissipation rate occurs close to the front of the particle surface due to the increased strain rates (both extensional and compressive) as the particles move through the surrounding turbulent eddies. For fixed volume fraction and diameter of the particles, the most pronounced effects on TKE, its dissipation rate and its rate of change due to two-way coupling occur by increasing the ratio ρ_p/ρ_f which is directly proportional to the Stokes number, (τ_p/τ_k), and the particles mass fraction, ϕ_m.

9:31AM GL.00008 Mixing of inertial particles at a turbulent - non turbulent interface , SERGIY GERASHCHENKO, Cornell University, LUIS RUELAS, San Jose State University, ZELLMAN WARHAFT, Cornell University — Motivated by the problem of entrainment of dry air into clouds, water droplets are sprayed into the high turbulence side of a shearless turbulence mixing layer: a layer in which there is a step in turbulence intensity across the interface but there is negligible change in the mean velocity (Veeravelli and Warhaft, JFM, 1989, 208, 191). Active and passive grids are used to form the mixing layer. A splitter plate is used to separate droplet-non droplet interface near the origin. Particle concentration, size and velocity are determined by Phase Doppler Particle Analyzer, the velocity field by hot wires, and the droplet accelerations by particle tracking. The results are compared with injecting the particles into one side of homogeneous turbulence. We show that the particle number density is approximately an order of magnitude smaller on the low turbulence side of the turbulent-non-turbulent interface compared with that of a turbulent- turbulent interface with the same initial distribution of inertial particles on one side. Stokes and Froude number effects are investigated. Sponsored by the U.S. NSF.

9:44AM GL.00009 Large scale accumulation of inertial particles in turbulent channel flow , LUCA BRANDT, PHILIPP SCHLATTER, Linne Flow Centre, KTH Mechanics, Stockholm, GAETANO SARDINA, CARLO M. CASCIOLA, Department of Mechanics and Aeronautics,University of Rome "La Sapienza" — Spatially inhomogeneous turbulent flows induce peculiar phenomena on the transport of a dispersed phase of inertial particles. In channel flows the most striking effect is the spatial segregation of particles that may achieve a concentration at the wall largely exceeding that in the bulk. Here we approach the issue by considering direct numerical simulations in a channel seeded with different populations of diluted, tiny particles. The simulations at Re_τ=180 have been performed using the largest domain size so far. The structures found in the fully developed stage of the process show strong spanwise correlations more intense than those found in the corresponding elongated structures of low and high fluid speed. The extremely regular spanwise organization corresponds to a mean spacing of about 120 plus units. The turbulent simulations with an increased size of the numerical box highlight some significant differences in the correlation of particle concentrations. A possible explanation of this feature can be related to large-scale structures of the velocity field, which might carry a considerable amount of energy. Correlations between turbulent events, sweep and ejections, and the particle motion to and from the wall will be also presented.

9:57AM GL.00010 Self-similarity in particle laden flows at constant volume[1] , MATTHEW MATA, UCLA, NATALIE GRUNEWALD, Univ. Bonn, RACHEL LEVY, Harvey Mudd College, THOMAS WARD, NCSU, ANDREA BERTOZZI, UCLA — We consider constant volume thin film slurries on an incline. Clear fluids in this geometry are known to have a front position that moves according to a $t^{1/3}$ scaling law, based on similarity solution analysis [Huppert, Nature, 1982]. We investigate the same dynamics for particle laden flow using a recently proposed lubrication model for the slurry and physical experiments. Our analysis includes the role of a precursor in the model. We conclude that in the lubrication model, the height of the precursor significantly influences the speed of the fluid front, independent of particles settling in the direction of flow. By comparing theory with experiments we conclude that the $t^{1/3}$ scaling law persists, to leading order, for slurry flows with particle settling. However additional physics is needed in the existing lubrication models to quantitatively explain departures from clear-fluid self-similarity due to particle settling.

[1]This research was supported by ONR grant N000140610059, NSF grants ACI-0321917, DMS-0601395, and DMS-0502315, the UC Lab Fees Research Program, and DFG grant GR 3391/1-1.

Monday, November 23, 2009 8:00AM - 9:44AM –

Session GM Supersonic/Hypersonic I 200B

8:00AM GM.00001 Fluid-structure interaction of converging shocks in water[1] , VERONICA ELIASSON, California Institute of Technology, WILLIAM D. HENSHAW, Lawrence Livermore National Laboratory, ARES J. ROSAKIS, PAUL E. DIMOTAKIS, California Institute of Technology — Numerical simulations of shock focusing in a convergent water-filled geometry with various types of elastic solids (rubber, plastic and metal) as the surrounding material have been performed. The fluid deforms the solid, generating elastic waves, which in turn affect the liquid; thus creating a coupled fluid-structure problem. Here, we use the Overture suit, a code for solving partial differential equations on curvilinear overlapping grids using adaptive mesh refinement. The Euler equations with a stiffened equation of state are used in the fluid domain and linear elasticity is assumed in the solid domain. Preliminary results indicate that the wave speed of the material has a significant influence on the behavior of the converging shock. Comparisons between numerical and experimental results are presented and have the potential to aid in the design of marine structures with convergent sections subjected to dynamic loading events.

[1]Supported by ONR.

8:13AM GM.00002 Mixing analysis of PLIF images in a multi-stream injection nozzle[1] , C. RANDALL TRUMAN, PETER VOROBIEFF, Univ. of New Mexico — We present quantitative analysis of image sequences of multi-stream injection nozzle flows with several different injection geometries. Image sequences were acquired by A.M. Ragheb and G.S. Elliott (UIUC) using planar laser-induced fluorescence (PLIF) in iodine to visualize flow mixing. The injection nozzle was comprised of a slot, ejector and injector block, with rows of ejector and injector holes along the slot length. The ejector flow exits in an underexpanded state so that upon expanding it forces the slot and injector flows together to enhance mixing. For this study, the diameter and geometry of ejector holes were varied to assess their effect on mixing. Two configurations of ejector holes were used, each with two different diameters for a total of 4 cases with data collected at downstream stations. We carried out a quantitative mixing analysis for these configurations, using two parameters to quantify the mixing. The first parameter, the mixing quality criterion, is assessed from the statistics of the PLIF image intensity histograms, which are bimodal for poorly-mixed flows and have a single peak in well-mixed flows. The second parameter is mixing interface length. Our analysis shows that one injection scheme significantly enhances mixing by stretching the mixing interface.

[1]sponsor: Missile Defense Agency Phase II SBIR to CU Aerospace

8:26AM GM.00003 Analysis of DNS database of canonical shock/turbulence interaction , JOHAN LARSSON, Stanford University — A set of databases generated by direct numerical simulation of isotropic turbulence passing through a shock wave is analyzed. Averages conditioned on the local instantaneous shock strength are used to elucidate the structure of the shock/turbulence interaction through the strongest and weakest points on the shock. For sufficiently strong turbulence there exists completely smooth profiles through the shock-region. The unsteady shock-motion is analyzed and linked to the incoming turbulence.

8:39AM GM.00004 A Derivation of New Regularized Euler Equations from Basic Principles[1] , 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 wavemodes results in energy cascade to ever smaller scales in turbulence and 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. The author's group recently reported a regularization technique for the Burgers equation (Norgard and Mohseni 2008) which is now extended to one-dimensional compressible Euler equations (Norgard and Mohseni 2009). This investigation presents a formal derivation of these equations from basic principles. We will extend our previous results to multidimensional compressible and incompressible Euler equations. We expect this technique to simultaneously regularize shocks and turbulence. Numerical simulation demonstrating the shock regularization properties of these equations will be presented.

[1]The author acknowledges financial support from the AFOSR.

8:52AM GM.00005 A low-dissipation and dispersion finite volume method for large eddy simulation of compressible flow on arbitrary unstructured grids , FRANK HAM, Stanford University — One way to develop stable solvers for large eddy simulation with minimal numerical dissipation is to use so-called summation-by-parts (SBP) operators. These discrete operators mimic the integration-by-parts property of the continuous equations, leading to discrete stability by the energy method. Unfortunately, the application of large eddy simulation to compressible flows of engineering interest often involves complex geometries and consequently unstructured grids. In the present work, a method for constructing a fast, explicit compressible flow solver for large eddy simulation is presented that uses standard polynomial reconstruction techniques to build accurate finite volume operators, which are subsequently modified based on the extent to which they are not SBP. Because the SBP property is a property of the operators and not the solution, this can be performed as a pre-processing step in a large eddy simulation. Several examples will be presented demonstrating the robustness and accuracy of this compromise approach, including sound radiated from transonic and supersonic jets.

9:05AM GM.00006 High Resolution Direct Numerical Simulations of Compressible Isotropic Turbulence , MARK PETERSEN, DANIEL LIVESCU, JAMALUDIN MOHD-YUSOF, SUMNER DEAN, Los Alamos National Laboratory — We present results from a systematic study of Direct Numerical Simulations of forced compressible turbulence. The simulations explore the M_t–χ parameter space where M_t, the turbulent Mach number, varies from 0.02–0.6, and χ, the ratio of dilatational to solenoidal energy, varies from 0.01–10, on up to 1024^3 meshes, with maximum Taylor Reynolds numbers of $R_\lambda > 300$. Thus, the study covers the weakly to moderate compressibility effects regime as reflected in the turbulent Mach number values, as well as the low to strong dilatational effects regime that may arise independently from the Mach number effects, e.g. due to exothermic reactions. The forcing method is designed to control the statistically stationary state values of the dissipation (thus the Kolmogorov scale) and the ratio of dilatational to solenoidal dissipation. This ensures that the simulations are both stable and well resolved. The DNS results are used to examine the spectral properties of the solenoidal and dilatational velocity fields and highlight changes in the turbulence properties due to compressibility and dilatational effects.

9:18AM GM.00007 Development of a numerical code for the study of a supersonic planar wake , JEAN-PIERRE HICKEY, XIAOHUA WU, Royal Military College of Canada — The fully-developed supersonic planar wake represents a canonical high-speed flow occurring in many aeronautical applications. The goal of the current research program is to perform a high-quality direct numerical simulation in order to thoroughly compare the statistics with classical experimental data and gain a better understanding of the structures present in the far-field of a supersonic planar wake. In order to study this flow a code is under development using a very efficient modified MacCormack-type scheme to solve the governing equation set. The main drawback of this numerical method is the large dispersive errors occurring in regions of sharp gradients which can occur in as shocklets in highly compressible flow. To this effect, a study of the numerical properties of this scheme is done using classical one-dimensional test cases such as the Shu-Osher and the Sod problem. The scheme compares very favorably to typical compressible schemes such as the Pade and Roe solvers but shows a very significant advantage in terms of memory usage and speed.

9:31AM GM.00008 Effects of Turbulence on Taylor-Sedov Blast Waves in Radially-Symmetric Geometries[1] , TIBERIUS MORAN-LOPEZ, DANIEL ZAIDE, JAMES HOLLOWAY, University of Michigan, OLEG SCHILLING, Lawrence Livermore National Laboratory — Progress in extending studies of the classical Taylor–Sedov blast wave problem by incorporating effects due to turbulence is reported. Investigations consist of the analytical development and initial numerical findings describing the evolution of large and instantaneous energy releases from point explosions (in radially-symmetric systems) while coupling turbulent instabilities. The closure of the Reynolds-Favre averaged mean flow equations is accomplished using a $K-\epsilon$ model in the gradient diffusion approximation. To reduce the complexity of the problem, self-similar analysis is used to reduce the space-time dependent system of partial differential equations to coupled, nonlinear ordinary differential equations in the self-similarity variable. Preliminary approximations considered in the problem are also discussed.

[1]This work was performed under the auspices of Lawrence Livermore National Laboratory under Contract No. DE-C52-07NA27344. This work was also partially supported by the DOE NNSA under the Predictive Science Academic Alliances Program by grant DE-FC52-08NA2

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GN Non-Newtonian Flows I 200C

8:00AM GN.00001 Analytical solution for creeping channel flow of non-Newtonian compressible fluid subject to wall slip , HANSONG TANG, Dept. of Civil Eng., City College, City Univ. of New York, DIHAN KALYON, Highly Filled Material Inst., Stevens Inst Tech — Creeping channel flows of compressible viscoplastic fluids subject to wall slip are important in many industries as well as presenting significant academic challenges. Here we present analytical solutions for pressure-driven steady flows of viscoplastic fluids within planar and circular channels. Herschel-Bulkley constitutive equation is employed in conjunction with constant or pressure dependent wall slip coefficients. Simplifications of the Hershel-Bulkley fluid provide other generalized Newtonian fluids including power law and Newtonian fluids. Under the assumption that pressure only changes in the flow direction and its gradient deviates slightly from a constant, explicit solutions are derived for distributions of pressure, velocity, and slip velocity within the channels. The analytical solutions are compared against numerical solutions as well as experimental data collected using rectangular slit dies. The effects of compressibility and wall slip on the flows are elucidated. A distinctive feature of such a flow is that, when the slip coefficient is considered to be inversely proportional to pressure, the slip velocity increases rapidly in the flow direction and the flow can evolve into a pure plug flow at the exit, removing the stress singularity that is presumed to exist in the transition from the channel flow into a free surface flow at the exit.

8:13AM GN.00002 Transient growth without inertia , MIHAILO R. JOVANOVIĆ, SATISH KUMAR, University of Minnesota — We study transient growth in inertialess plane Couette and Poiseuille flows of viscoelastic fluids. For streamwise-constant 3D fluctuations, we demonstrate analytically the existence of initial conditions that lead to quadratic scaling of both the kinetic energy density and the elastic energy with the Weissenberg number, We. This shows that in strongly elastic channel flows, both velocity and polymer stress fluctuations can exhibit significant transient growth even in the absence of inertia. Our analysis identifies the spatial structure of the initial conditions (i.e., components of the polymer stress tensor at $t = 0$) responsible for this large transient growth. Furthermore, we show that the fluctuations in streamwise velocity and the streamwise component of the polymer stress tensor achieve $O(We)$ and $O(We^2)$ growth, respectively, over a time scale $O(We)$ before eventual asymptotic decay. We also demonstrate that the large transient responses originate from the stretching of polymer stress fluctuations by a background shear and draw parallels between streamwise-constant inertial flows of Newtonian fluids and streamwise-constant creeping flows of viscoelastic fluids. One of the main messages of this work is that, at the level of velocity fluctuation dynamics, polymer stretching and the Weissenberg number in elasticity-dominated flows effectively assume the role of vortex tilting and the Reynolds number in inertia-dominated flows of Newtonian fluids.

8:26AM GN.00003 Extensional dynamics of viscoplastic filaments , ANJA SLIM, NEIL BALMFORTH, NEVILLE DUBASH — A one-dimensional slender-thread model is used to explore viscoplastic dripping under gravity and the controlled extension of a liquid bridge. We describe dynamics up to pinch-off and consider the possibility of using measurements in the two configurations (eg. drop volume) to infer rheological parameters. The model results are compared with experiments using aqueous solutions of Carbopol and kaolin suspensions.

8:39AM GN.00004 Multiscale simulation of polymer melt flows between parallel plates , SHUGO YASUDA, RYOICHI YAMAMOTO, Dept. of Chemical Engineering, Kyoto University — The behaviors of polymer melt composed of short chains with ten beads in parallel plates are simulated by using a hybrid method of molecular dynamics and computational fluid dynamics. The creep motion under a constant shear stress and recovery after removing the stress, the pressure driven flows and the flows in rapidly oscillating plates are simulated. The flow profiles of polymer melt are quite different from those of the Newtonian fluid due to the elasticity or the shear thinning. The delayed elastic deformation and plug-like velocity profile are reproduced, respectively, in the creep and pressure driven flow. In the rapidly oscillating plates the viscous boundary layer of the melt is much thinner than that of the Newtonian fluid due to the shear thinning of the melt. Three different rheological regimes, i.e., the viscous fluid, viscoelastic liquid, and viscoelastic solid regimes, form over the oscillating plate according to the local Deborah numbers. The melt behaves as a viscous fluid in a region for $\omega\tau^R \lesssim 1$, and the crossover between the liquid-like and solid-like regime takes place around $\omega\tau^\alpha \simeq 1$ (where ω is the angular frequency of the plate and τ^R and τ^α are Rouse and α relaxation time, respectively).

8:52AM GN.00005 Experimental study of the hydrodynamic interaction between a pair of bubbles ascending in a non-Newtonian liquid , DIEGO SAMANO, RODRIGO VELEZ, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — We present some experimental results about the interaction of a pair of bubbles ascending in non-Newtonian fluids. A high speed camera was used to follow in-line and off-line rising motion of two bubbles in a Newtonian fluid (a glycerin-water solution), a Boger fluid (aqueous polyacrylamide solution), and a shear-thinning fluid (aqueous xanthan solution). For the case of shear-thinning fluids, the power index, n, affects the tendency of the bubble pair to aggregate. Therefore, in addition to bubble separation, orientation and Reynolds number, the hydrodynamic force depends strongly on the shear-thinning nature of the fluid. Several examples will be shown. For elastic fluids, the Deborah number affects the hydrodynamic interaction. We found that the appearance of the negative wake changes the nature of the interaction substantially. Some examples and comparisons with numerical results will be presented.

9:05AM GN.00006 A numerical study of the hydrodynamic interaction of bubble pairs ascending in non-Newtonian liquids , RODRIGO VELEZ, Universidad Nacional Autonoma de Mexico, PENGTAO YUE, Virginia Polytechnic Institute and State University , JAMES J. FENG, University of British Columbia, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — This talk presents computational results on the interaction of a pair of bubbles immersed in non-Newtonian fluids. The Arbitrary Lagrangian-Eulerian (ALE) technique was used to simulate two bubbles rising in tandem or side by side in shear-thinning and Oldroyd-B fluids. In the shear-thinning fluid, the pairwise interaction is affected by the the Eotvos and Reynolds numbers as well as the initial orientation of two bubbles. In particular, two in-line bubbles will rise together and form a doublet as the trailing bubble catches up with the leading one. In a viscoelastic fluid, a negative wake may appear depending on the initial separation between the bubbles. The capillary number, which can be an indicator of the bubble deformability, seems to play a secondary role in the bubble interaction. The numerical simulations complement previous experiments done with bubble swarms by our group.

9:18AM GN.00007 Selective withdrawal of non-Newtonian fluids: surface deformation induced by a sink flow , DIWEN ZHOU, JAMES FENG, Department of Chemical and Biological Engineering, University of British Columbia — This talk reports experiment and numerical studies of selective withdrawal in a fluid-gas system. Using visual observation and finite element simulations based on an Arbitrary Lagrangian-Eulerian scheme, we have explored the effects of viscoelasticity on the deformation of free surface when the fluid is polymer solution (experiment) or Giesekus fluid (simulation). In the experiments, we find a thin air jet emanating from the tip of the free surface for polymer solutions when the distance between the free surface and the sink is below a critical value. This does not occur for the free surface of Newtonian liquids, and is caused by the additional elongational stress due to the polymer. In the simulations, the effects of elasticity on the surface deformation have been captured. The balance between surface and viscoelastic forces may potential be used for measuring extensional viscosity.

9:31AM GN.00008 Active and hibernating turbulence in minimal channel flow of Newtonian and polymeric fluids , LI XI, MICHAEL GRAHAM, University of Wisconsin-Madison — The experimental observation of minute amount of polymers reducing turbulent drag has been long established. In this study, we isolate the turbulent self-sustaining process by conducting direct numerical simulations (DNS) in minimal flow units (MFU). These solutions obtained at various polymer parameters recover all key transitions in viscoelastic turbulent flows reported previously in experiments at much higher Re, including the onset of drag reduction, low degree of DR (LDR), high degree of DR (HDR) and maximum drag reduction (MDR). At MDR, the mean velocity profile is insensitive to changing polymer parameters. The LDR-HDR transition is characterized by a sudden increase in the minimal box size of sustaining turbulence, which may correspond to a qualitative change in the self-sustaining mechanism. Dynamics of turbulence show intermittent appearance of "hibernation" periods, which are characterized by long-lasting flow structures with low instantaneous wall shear stress and low turbulence intensity. These periods appear both in Newtonian and viscoelastic fluids; however they are observed much more frequently in HDR and MDR stages, which contribute substantially to the relatively high level of DR. Instantaneous velocity profiles during hibernation periods resemble the Virk MDR profile, including the disappearance of the log-law layer and a comparable slope with the Virk MDR asymptote.

9:44AM GN.00009 Entangled chain dynamics of polymer knots in extensional flow , LOUISE WILKIN, DEMOSTHENES KIVOTIDES, THEO THEOFANOUS, University of California, Santa Barbara — We formulate a coarse grained molecular dynamics model of polymer chains in solution that includes hydrodynamic interactions, thermal fluctuations, nonlinear elasticity, and topology-preserving solvent mediated excluded volume interactions. The latter involve a combination of potential forces with explicit geometric detection and tracking of chain entanglements. By solving this model with numerical and computational methods, we study the physics of polymer knots in strong extensional flow (Deborah number, $De = 1.6$). We show that knots slow down the stretching of individual polymers by obstructing via entanglements the "natural", unraveling, flow-induced chain motions. Moreover, the steady state polymer length and polymer-induced stress values are smaller in knotted chains than in topologically trivial chains. We indicate the molecular processes via which the rate of knot tightening affects the rheology of the solution.

9:57AM GN.00010 A Finite Volume Solver for Non-Newtonian flow on Unstructured Grid with Application in Blood Flow , GAOLING ZHOU, BIN CHEN, State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an — In order to simulate blood flow in complex vessel, a finite volume solver for Casson fluid flow based on SIMPLE algorithm of Newtonian fluid on unstructured collocated grid is developed. For the discretization of convective fluxes and source term, it is similar with Newtonian fluid. For the discretization of diffusion fluxes, viscosity will take the value calculated from the flow field of previous iteration in order to avoid the complexity caused by the complicated viscosity expression as a function of shear rate. Then the discretization of momentum equation is similar with that of Newtonian fluid with variable viscosity and SIMPLE algorithm can be used to resolve the pressure-velocity coupling. Casson fluid flow through a symmetric sudden expansion channel is compared with literature and the good agreement between simulated velocity distributions with literature confirms the validation of present algorithm. Afterwards, blood flows in T-type bifurcation are simulated by our proposed algorithm. The simulation result of Casson fluid is more consistent with experiment than that of Newtonian fluid, which indicates that using Casson model to simulate on-Newtonian characteristics of blood is successful and necessary.

Monday, November 23, 2009 8:00AM - 9:57AM –
Session GP Instability: Jets & Wakes III 200D

8:00AM GP.00001 Visualization of Pulsating Low-Speed Flows from a Basic Annular Jet[1] , A. SANTIAGO PADRON, University of Minnesota, JOHN BAKER, University of Alabama — Results of a study involving pulsating low-speed free jets issuing from an annular orifice into a quiescent medium are discussed. Transient flow behavior associated with pulsating jets is known to affect entrainment, mixing, and spread rate characteristics. Also, annular jet flows often provide a better description of the flow associated with nozzles used in engineering applications. However, the flow phenomena related to pulsating annular jets is still not fully understood. In this study, flow in the initial region of a pulsating low-speed annular water jet issuing into a quiescent water reservoir was visualized by means of a dye. The blocking ratio was fixed at 0.7. The Reynolds number was varied from 59 to 155 and the Strouhal number from 0.133 to 1.90. For the experimental conditions considered, two different flow regimes were observed. At high pulse frequencies, the flow field resembled that of the steady annular jet. As the frequency decreased, the flow transitions into a structure composed of a train of toroidal vortices, i.e. vortex rings. The frequency at which transition occurred was proportional to the Reynolds number.

[1]Work performed under REU site sponsored by NSF grant EEC0754117.

8:13AM GP.00002 The stability of multiple tip vortices , JIM DENIER, JAMES WHITEHEAD, The University of Adelaide — The talk will present some new results on the viscous stability of multiple tip vortices. Using a BiGlobal stability approach we predict the critical Reynolds number for a variety of vortex configurations. Some discussion of the inviscid instability of these multiple vortex configurations will also be given.

8:26AM GP.00003 Evolution of turbulent jets in low aspect ratio containers[1] , S. POL, C. NATH, D. GEST, S. VOROPAYEV, H.J.S. FERNANDO, Arizona State University, S. WEBB, Sandia National Labratories — The evolution of homogeneous and buoyant turbulent jets released into a low aspect ratio (width/height) container was investigated experimentally using PIV, MSCT probing and digital imaging. The motivation was to understand mixing process occurring in U.S. Strategic Petroleum Reserves (SPR), where crude oil is stored in salt caverns of low aspect ratio. During maintenance or filling, oil is introduced as a jet from the top of the caverns. This study is focussed on mean and turbulent flow characteristics as well as global flow instability and periodic oscillations intrinsic to jets in low aspect ratio containers. Scaling arguments were advanced for salient flow parameters, which included the characteristic length (container width D) and velocity (for homogeneous jets, $J^{1/2}D$, where J is the momentum flux at the jet exit) scales. For buoyant jets, the buoyancy flux B needs to be introduced as an additional parameter. Such jet flows do not reach a steady state, but bifurcate periodically with a frequency scale $J^{1/2}/D^2$ while enhancing global mixing.

[1]Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company for the NNSA under contract DE-AC04-94AL85000.

8:39AM GP.00004 Controlling a liquid jet inside the regular breakup regime by applying a composite disturbance to the actuator , MINA ROHANI, DEREK DUNN-RANKIN, FARYAR JABBARI, University of California, Irvine — In this work, we control the breakup characteristics of a liquid jet by manipulating a piezoelectric actuator and thus the disturbance applied to the jet. We are thereby able to provide desirable droplet size patterns over a wide frequency range. The regular breakup regime refers to the frequency range where the breakup characteristics are repeatable. Thus, although the droplets may not be uniform in size, they pinch off the stream at a constant rate. The regular breakup regime for different jet velocities and diameters has been specified experimentally. The experiments show formation of secondary droplets between main droplets, mostly around the lower frequency range of the regular breakup regime. We remove these secondary droplets by sending a composite disturbance comprising of a fundamental disturbance at the principal driving frequency and another harmonic mode. The choice of the additional harmonics depends on the desired droplets size pattern. It is thus possible to relate initial input disturbance waveform to the droplet formation pattern.

8:52AM GP.00005 Three-dimensional radiative instabilities in a stratified plane jet , JULIEN CANDELIER, CHRISTOPHE MILLET, CEA, DAM, DIF, F-91297 Arpajon, France, STÉPHANE LE DIZÈS, IRPHE, CNRS, BP 146, F-13384 Marseille, France — We investigate the three-dimensional stability of a stratified plane Bickley jet in the Boussinesq approximation framework. The angle θ between the shear plane and the direction of stratification and the Froud number Fr are considered as a control parameters. Following the parallel flow approximation, the instability wave solution is sought in the form of a normal mode in two directions. We draw attention to a mechanism whereby Kelvin-Helmholtz mode may have a radiative structure and more generally how internal waves (or gravity waves) associated with unstable radiative modes may be spontaneously generated.

9:05AM GP.00006 Microphone-array measurements of the surface-pressure field produced by oblique and normal impinging jets[1] , A.M. NAGUIB, W. JIANG, K. ZHANG, Michigan State University, East Lansing, MI-48824, USA, M. EL-ANWAR, A.M. ABOUEL-FOTOUH, National Research Center, Giza, Egypt — The sptio-temporal, wall-pressure fluctuation generated by an axi-symmetric jet impinging on a flat wall is measured using a 30-microphone array. The focus of the study is the influence of the impingement angle on the strength, spatial distribution and space-time characteristics of the unsteady wall-pressure field. The investigation is conducted at jet Reynolds number of approximately 13000, based on jet diameter and three impingement angles: 0, 15 and 30 degrees. The results show that the impingement angle has strong influence on the level of pressure fluctuations, leading to large increase on the side where the flow experiences less turning (relative to normal impingement), and vice versa. Substantial influences are also found on the spatial characteristics and convection velocity of the pressure-generating flow structures. These effects and others will be presented and discussed in this talk.

[1]This work is funded by US-Egypt Joint Board on Scientific and Technological Cooperation through contract no. OTH10-028-002 and NSF grant number OISE-0611984.

9:18AM GP.00007 Primary Breakup of a High Speed Liquid Jet , WURIGEN BO, XINGTAO LIU, JAMES GLIMM, Stony Brook University — The primary breakup of a high speed jet is studied numerically in 2D and 3D using the front tracking method. We introduce an improved, robust, locally grid based method for reconstruction of tangled interfaces. This method improves the handling of topological change of the surface mesh in the 3D simulations, and is essential for the success of the simulations presented here. From the 2D axisymmetric simulations, we find agreement with experiment in regard to the tip velocity of the jet and its overall degree of breakup or spreading. Due to resolution restrictions, we observe in 3D breakup primarily in the jet tip region and somewhat larger droplets than expected from theory.

9:31AM GP.00008 Forming a fine jet in inkjet printing , E.Q. LI, J.Y.H. FUH, Y.S. WONG, Department of Mechanical Engineering, National University of Singapore, S.T. THORODDSEN, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia — The formation of fine jets during the piezoelectric drop-on-demand inkjet printing has been investigated using ultra-high-speed video imaging. The speed of the jet can exceed 80 m/s, which is much higher than the general drop velocity during inkjet printing. The diameters of the thinnest jets are of the order of a few microns. The generation of such fine jets has been studied over a wide range of viscosities, using 7 different concentrations of water-glycerin solutions. This jetting is associated with the collapse of an air-pocket which is sucked into the nozzle during the printing. This occurs for longer expansion times for the piezo-element. We have characterized the relationship between the speed of the fine-jet and other parameters like the diameter of the jet and the physical properties of the liquid.

9:44AM GP.00009 Saturation of the Magnetorotational Instability at Large Elssaser Number , KEITH JULIEN, BENJAMIN JAMROZ, University of Colorado at Boulder, EDGAR KNOBLOCH, University of California at Berkeley — The MRI is believed to play an important role in accretion disk physics in extracting angular momentum from the disk and allowing accretion to take place. The instability is investigated within the shearing box approximation under conditions of fundamental importance to astrophysical accretion disk theory. The shear is taken to be the dominant source of energy, but the instability itself requires the presence of a weaker vertical magnetic field. Dissipative effects are suffiently weak that the Elsasser number is large. Thus dissipative forces do not play a role in the leading order linear instability mechanism. However, they are sufficiently large to permit a nonlinear feedback mechanism whereby the turbulent stresses generated by the MRI act on and modify the local background shear in the angular velocity profile. To date this response has been omitted in shearing box simulations and is captured by a reduced pde model derived from the global MHD fluid equations using multiscale asymptotic perturbation theory. Results from simulations of the model indicate a linear phase of exponential growth followed by a nonlinear adjustment to algebraic growth and decay in the fluctuating quantities. Remarkably, the velocity and magnetic field correlations associated with these growth and decay laws conspire to achieve saturation of angular momentum transport.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GQ Instability: Interfacial and Thin-Film IV 200E

8:00AM GQ.00001 Fingering Instability During Debonding: From a Viscous Liquid to a Soft Elastic Solid , JULIA NASE, PPMD-ESPCI, ANKE LINDNER, PMMH-ESPCI, COSTANTINO CRETON, PPMD-ESPCI — We investigate the fingering instability during debonding of a confined viscoelastic layer in a circular lifted Hele-Shaw cell.[1] We use PDMS with different degrees of crosslinking, ensuring a continuous transition from a viscous liquid to a soft elastic solid. During debonding, a fingering instability with characteristic initial wavelength λ evolves. When going from a liquid to a solid, we observe a transition from bulk to interfacial mechanisms. We predict this transition from linear viscoelastic and surface properties. We show that for the interfacial mechanism, λ depends solely on the film thickness, whereas for the bulk mechanism, λ depends on the material parameters. $lambda$ is in both cases in quantitative agreement with linear stability analysis. For a Newtonian oil, we discuss in detail the coarsening of the pattern during debonding. Adapting a recent 3D technique, we visualize for the first time in situ the contact line between viscoelastic material and air in three dimensions, providing direct access to the boundary conditions.

[1]J. Nase, A. Lindner, C. Creton, PRL **101**, 074503 (2008)

8:13AM GQ.00002 The effects of viscosity and pressure on the bursting of a drop in a Hele-Shaw cell[1] , ANDREW WHITE, THOMAS WARD, North Carolina State University — As one fluid is injected into another fluid of greater viscosity, instabilities occur in the form of fingers which extend radially from the injection point (Saffman & Taylor, Proc. R. Soc. Lon. A, 1958). As the lower-viscosity fluid reaches the free surface it rapidly bursts through the higher- viscosity fluid (times are typically less that 50 ms for our system) and a pressure drop occurs. This pressure drop induces the shrinking of the non-bursting fingers. By varying the air pressure and water-glycerol viscosity we study this process by analyzing sequences of images prior and after the bursting event inside a Hele-Shaw cell with a gap spacing of between 10 and 500 micrometers. It has been shown that in a micro-scale environment the effects of gravity are negligible as fluid flow is dominated by capillary forces, thus such a setup would behave in space just as it does on Earth. Therefore it may then be possible to use hot air injected into a Hele-Shaw cell filled with water to generate steam in a microgravity environment.

[1]NC Space Grant

8:26AM GQ.00003 Two-phase hydrodynamic model for air entrainment at moving contact line , TAK SHING CHAN, JACCO SNOEIJER, Physics of Fluids Group, Department of Science and Technology, University of Twente, the Netherlands — The moving contact line problems are challenging because they involve multiple length scales. One interesting case arises when an advancing liquid of high viscosity entrains the surrounding phase, such as air. In this presentation, we introduce a hydrodynamic model that generalizes the lubrication theory in order to take into account the velocity fields of the two phases. Assuming that the curvature of the interface is small we derive a differential equation for the interface profile at stationary state. We found that there is a critical capillary number above which no steady meniscus can exist and instability will occur. For example, air bubbles will be entrained into the liquid at the advancing contact line. However, we found no instability when neglecting the viscosity of the surrounding phase, illustrating the two-phase nature of the problem.

8:39AM GQ.00004 Surfactant effect on the motion of long bubbles in capillary tubes , PRABIR DARIPA, Texas A&M University — In this talk, we give a theoretical proof of the thickening effect of surfactant by considering a small concentration of surfactant Γ and variable surface tension on a long bubble interface which is moving slowly and steadily in a capillary tube filled with a liquid of viscosity μ. The contact angle is taken as zero at the walls and the gravitational effect is neglected. This problem was originally considered by Bretherton and later studied numerically by Park (1990) and Ratulowski and Chang (1991). The main result we obtain is a formula of the film thickness in terms of M and Γ where M is the Marangoni number. A comparison with Bretherton's "clean" case shows the thickening effect of surfactant. This talk is partially based on an going work with Gelu Pasa.

8:52AM GQ.00005 Stability of liquid sheet edges , ROUSLAN KRECHETNIKOV, UCSB — Accelerating edges of thin liquid sheets are ubiquitous and are known to experience a longitudinal (along-the-edge) instability, which often leads to their break-up and atomization. The fundamental physical mechanisms of this instability are studied analytically in the form of a concise model. It is discovered that the classical Rayleigh-Taylor mechanism is substantially modified which leads to a stability picture different from that for flat interfaces, in part due to an interplay with Rayleigh-Plateau mechanisms. In particular, as the Bond number increases, first only one critical wave number is excited, but for higher values of the Bond number several critical wavenumbers can coexist with the same growth rates. This allows for the transition from the regular picture, in which one wavelength sets the pattern, to the frustrated picture, in which a few wavenumbers compete with each other.

9:05AM GQ.00006 A Phase-field Model of Wetting in Porous Media – Origin of Gravity Fingering During Infiltration , RUBEN JUANES, LUIS CUETO-FELGUEROSO, MIT — We present a new continuum mathematical model of wetting into dry soil. The inspiration for the new model is the flow of thin films (like water down a plane), which also displays fingering instability. The key idea is very simple: the macroscopic equations must reflect the presence of a macroscopic interface—the wetting front. We then cast the model in the rigorous framework of phase-field models and nonlocal thermodynamics. The new model is appealing. It is a simple extension of the traditional model—Richards' equation—with a new term (a fourth-order derivative in space) but without any new parameters. It reproduces the two key features of unsaturated flow: a nonmonotonic saturation profile, and gravity fingering. It explains why, when, and how, fingers form. It shows excellent quantitative agreement with experiments in terms of tip saturation, tip velocity and finger width. The most attractive aspect is, however, that the new model offers a starting point for fundamentally new formulations of multiphase flow in porous media.

9:18AM GQ.00007 Transient numerical simulation of miscible channel flow with heat transfer and viscous heating[1] , KIRTI SAHU, PRASHANT VALLURI, Imperial College London, HANG DING, University of California, Santa Barbara, OMAR MATAR, Imperial College London — Pressure-driven miscible channel flow undergoing heat transfer and viscous heating, focusing on the displacement of a highly viscous fluid by a less viscous one, is studied by direct numerical simulations using the finite volume method. The flow dynamics are governed by the continuity and Navier-Stokes equations, coupled to an energy equation and a convective- diffusion equation for the concentration of the more viscous fluid through a concentration- and temperature- dependent viscosity. The effect of temperature difference, Nahme, Prandtl, and Schmidt numbers on the propagation of the front separating the two fluids and temporal evolution of the mass of the less viscous fluid is examined.

[1]EPSRC Grant number EP/D503051/1 and TBS, UK.

9:31AM GQ.00008 Numerical simulations of two-fluid channel flow with wall deposition and ageing effects[1] , DANIELE SILERI, KIRTI SAHU, OMAR MATAR, Imperial College London — 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. A Cahn-Hilliard equation is also solved for the volume fraction. Numerical solutions are obtained subject to no-slip and no-penetration conditions at the walls, and constant flow rate conditions at the channel inlet; outflow conditions are imposed at the outlet. This model accounts for a thermal instability in the bulk, through a chemical equilibrium model based on the Gibbs free energy, which leads to the formation of the highly viscous phase and its deposition at the channel walls. We also account for the evolution of the rheology of the deposited phase through "ageing." We present results showing typical flow dynamics and the effect of system parameters on the average deposit thickness.

[1]EPSRC Grant number EP/D50351/1

9:44AM GQ.00009 Displacement flows between Newtonian fluids at moderate Reynolds numbers in rectangular channels , PRASHANT VALLURI, Imperial College London, HANG DING, University of California, Santa Barbara, PETER SPELT, OMAR MATAR, Imperial College London — Displacement flows between two Newtonian fluids in rectangular channels is studied by numerical and analytical means. Two-stages are clearly seen in the displacement process: first, a core of the displaced fluid is removed by a finger of the displacing fluid with width less than the channel height; then, the film of the displaced fluid adjacent to the wall left behind is then removed via interfacial instabilities that grow spatio-temporally. The shape of the finger dictated by a meniscus in the front and a tail of nearly asymptotic height; the latter is a function of the viscosity and density ratios, Weber number and Reynolds number. This dependence is studied by means of highly resolved direct numerical simulations using the diffuse-interface method. The interface shapes obtained is compared with analytical steady state solutions of the meniscus shapes in the downstream region and the asymptotic film thickness in the upstream region.

9:57AM GQ.00010 Effect of film thickness on EHD-driven instability of superimposed flows , PAYAM SHARIFI, ASGHAR ESMAEELI, Southern Illinois University at Carbondale — This study aims to investigate the effect of fluid layer thickness on EHD-driven instability of superimposed fluids using Direct Numerical Simulations (DNS). The geometric setup consists of two fluids having different electrical properties confined between two horizontal electrods, where the lighter fluid is overlaid on top of the heavier one. A front tracking/finite difference scheme is used, in conjunction with Taylor's leaky dielectric model, to solve the governing electrohydrodynamics equations in both fluids at finite Reynolds numbers and the dynamics of the interface and incipience of instability is investigated as a function of the thickness of the lower layer.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GR Viscous Flows I 200F

8:00AM GR.00001 Low Reynolds Flow Visualization Revisited: Free-Surface and Wall Effects , SHELLEY CHAN, JOSUE SZNITMAN, ALEXANDER SMITS, Princeton University — Many of the seminal experimental flow visualizations at low Reynolds number can be attributed to the pioneering works of S. Taneda. These classic investigations are still considered today benchmark visualizations and are widely used as textbook examples (Van Dyke, *An Album of Fluid Motion*, 1982). With the advent of modern quantitative flow visualization techniques, we are in a position to revisit in more detail some of the original questions posed by Taneda, including boundary effects on viscous flows surrounding objects (*J Phys Soc Jpn*, 1964). In the present talk, we conduct experimental flow visualizations around three-dimensional objects at low Reynolds number (Re=$O(10^{-3}$-$10^{-1})$). Quantitative visualizations are implemented in a tow tank using velocimetry measurements (PIV); models including cubes and spheres are submerged in a highly viscous Newtonian fluid (silicon oil, 5000x viscosity of water). Here, we discuss wall effects on velocity profiles in the near- and far-field surrounding such objects. Moreover, we interrogate the influence of the free surface of the tank on the resulting viscous flow fields. The present experimental setup offers a versatile framework to investigate a wide range of fundamental fluid mechanical problems relating flows at low Reynolds number.

8:13AM GR.00002 Transition from Hele-Shaw Flow to 2-D Creeping Flow , JOHN CIMBALA, Penn State, MANOOCHEHR KOOCHESFAHANI, Michigan State — In the Hele-Shaw experimental technique, liquid flows at very low Reynolds number through the narrow gap b between parallel plates. When a body is inserted between the plates, and dye is introduced upstream, the streaklines appear nearly identical to streamlines of steady 2-D potential flow over a body of the same shape. For example, Hele-Shaw flow does not separate at sharp corners, just like potential flow. However, if the plates are very far apart (large b), the resulting creeping flow at the same low Reynolds number is observed to separate at sharp corners, unlike potential flow. Here, we investigate how the flow changes from Hele-Shaw flow (small b) to 2-D creeping flow (large b). Low Reynolds number CFD simulations of a fence of height s along a wall in a channel reveal that the transition from Hele-Shaw flow to 2-D creeping flow is not sudden, but rather quite gradual as channel gap width is increased. Separation bubbles appear at small b/s, and grow in size as b/s increases. The reattachment length reaches 1% of the 2-D value at $b/s \approx 0.21$, but it does not reach 99% of the 2-D value until $b/s \approx 150$. Furthermore, for all values of b/s for which separation and reattachment are observed, even for large b/s (> 100), the reattachment length of the separation bubble is non-uniform across the span; it starts high, dips to a minimum, and then slowly rises, reaching 99% of the center plane value beyond about $15s$ to $20s$ from the wall.

8:26AM GR.00003 Three-dimensional corner flows in microchannels , LAURA GUGLIELMINI, Princeton University, ROBERTO RUSCONI, SIGOLENE LECUYER, Harvard University, HOWARD A. STONE, Princeton University — We study, by means of three-dimensional numerical simulations and analytical investigations, low Reynolds number fluid flows in rectangular micro-channels that present sharp angles, bends or curved boundaries. These flows are characterized by the generation of secondary streamwise vorticity, adjacent to the boundary, whose intensity is related to the rate of change of the curvature of the boundary (Balsa, 1998). We also study how this not well-known, yet relevant phenomenon affects the transport of scalar quantities at the boundary.

8:39AM GR.00004 Exchange flows in the low Reynolds number flow limit , LAURENCE RONGY, Harvard University, HOWARD STONE, Princeton University — We analyze the viscous gravity current that occurs when two fluids with different densities flow into each other in a two-dimensional channel. Assuming that the mixing between them and the surface tension at their interface are negligible, we study the flow within the lubrication approximation. For the general case of two fluids with different viscosities and in the presence of an imposed flow rate, the evolution of the current can be described by a single nonlinear PDE. When the mean flow rate is zero (closed channel) the model admits self-similar solutions for the thickness of the gravity current and solutions are obtained for different viscosity ratios. We also present numerical solutions for the gravity current in the cases of a non-zero imposed flow rate (open channel).

8:52AM GR.00005 Balancing a ball on a moving vertical wall covered in viscous fluid , TOM MULLIN, AAKESH DHATTA, University of Manchester — We present the results of experimental investigations into balancing heavy balls and cylinders on a vertical moving wall using a thin layer of viscous fluid. It is found that balance can be achieved over a very narrow range of speeds and the critical speed for fixed point behavior scales with the surface area of the cylinders and spheres. Surprising data collapse is achieved using the density of the particles.

9:05AM GR.00006 Visualization of Internal Flows with Pressure Oscillation and Surface Modification[1] , FELIX RIVERA, Inter-American University of Puerto Rico - Bayamon, JOHN BAKER, University of Alabama — A Stirling engine's displacer piston causes motion in its working fluid that exposes the fluid to pressure oscillations that directly impact flow behavior. Stirling engines are highly efficient external combustion engines that are often used in renewable energy applications and have been identified for use on near space platforms as auxiliary power units. The goal of this study was to identify the basic structures of the transient flow field within the expansion cylinder of a Stirling engine without the added complications introduced by convective heat transfer. A two-dimensional representation of the flow within the expansion cylinder of a Stirling engine was produced using an optically-accessible piston-enclosure configuration. The transient flow field within the enclosure was visualized using a rheoscopic fluid. The Reynolds number, based on the frequency of the piston oscillation and the stroke length, was varied from 1.74 to 9.05. Several transient flow structures are identified and the impact that an array of triangular fins has on these flow structures will be discussed.

[1]Work performed under REU site, NSF grant EEC 0754117, and NASA Award No. NNM07AA06A.

9:18AM GR.00007 Axisymmetric Ice Shelf Dynamics , SAM PEGLER, M.G. WORSTER, University of Cambridge — West Antarctica is composed principally of marine ice sheets, in which the mainland (grounded) ice sheet extends over the coastline as a floating ice shelf. Fed by snowfall far upstream, these sheets transport ice from the grounded component, over the grounding line, where the ice shelf lifts off, and into the ice shelf, which ultimately calves and adds water to the ocean. An idealized two-dimensional ice shelf has no dynamical influence on the grounded ice sheet or the position of the grounding line. However, horizontal stresses within a three-dimensional ice shelf, caused for example by ice rises or the lateral walls of a bay, can help fix the grounding line and prevent it from receding. This study investigates theoretically and experimentally the dynamics of an idealized three-dimensional ice shelf which flows radially from a point source, to elucidate the controlling influence of circumferential stresses within the shelf.

9:31AM GR.00008 A fully 3D experimental and theoretical study of flow patterns and Lagrangian trajectories generated by spinning bent rods in viscous fluids[1] , LONGHUA ZHAO, ELIZABETH BOUZARTH, PAVEL CHTCHEPROV, ROBERTO CAMASSA, DAVID MARRON, RICHARD MCLAUGHLIN, PETER PETROCHENKO, LEANDRA VICCI, UNC RTG FLUIDS GROUP, JOINT FLUIDS LAB, CAROLINA CENTER FOR INTERDISCIPLINARY APPLIED MATHEMATICS TEAM — The fluid motion induced by spinning cilia is fundamental to many living organisms. Under some circumstances it is appropriate to approximate cilia as rigid bent rods. We study the effects of shape and orientation of these idealized cilia upon flow structures in a Stokes fluid. By utilizing slender body theory and image method, an asymptotic solution is constructed for a slender body attached to a no-slip flat plane and rotating about its base sweeping out a cone. Using 3D stereoscopic projection in a table-top experiment we explore the complex flow structures and present quantified comparisons with the theoretical predictions. Intriguing short, intermediate and long time phenomena of particle trajectories are documented, and the intricacies of their theoretical modeling reported.

[1]Thanks support from RTG NSF DMS-0502266, NFS DMS-0308687 and NSF NIRT 0507151.

9:44AM GR.00009 A higher-order Hele-Shaw approximation for micro-channel flows , ITZCHAK FRANKEL, AMIR GAT, DANIEL WEIHS, Technion - Israel Institute of Technology — The classic hydrodynamic Hele-Shaw problem is revisited in the context of evaluating the viscous resistance to low-Mach compressible gas flows through shallow non-uniform microfluidic configurations (whose depths are small in comparison with all other characteristic dimensions). Earlier calculations have demonstrated that failure to satisfy the no-slip condition at the channel lateral walls severely restricts the applicability of the resulting approximation. To overcome this we have extended the calculation to incorporate an inner solution in the vicinity of the side walls (which, in turn, allows for the characterization of the effects of non-rectangular channel cross sections) and its matching to an outer correction. Comparison with finite-element simulations demonstrates a remarkably improved accuracy relative to the leading-order Hele-Shaw approximation. This suggests the present scheme as a useful alternative for the rapid performance estimate of microfluidic devices.

9:57AM GR.00010 Acoustic Droplet Vaporization through PDMS , DAVID LI, STANLEY SAMUEL, J. BRIAN FOWLKES, JOSEPH BULL, The University of Michigan — Acoustic droplet vaporization (ADV) involves the generation of bubbles from albumin-encapsulated perfluorocarbon (PFC) droplets that have been insonated with high intensity ultrasound (US). Gas embolotherapy, utilizing ADV, may facilitate occlusion of blood flow in the vasculature as bubbles undergo volume expansion of up to 125 times. Cancer therapy could benefit from such occlusions through starvation of the tumor. In order to visualize the detailed mechanics of vaporization and expansion process of the PFC droplets, idealized microvessels were constructed using polydimethylsiloxane (PDMS) channels. Microchannels (20 micron diameter) were fabricated using PDMS with polymer-crosslinker mixing ratios ranging from 5:1 to 20:1. Droplets were introduced into the channels and exposed to US for vaporization. Mixing ratios were observed to impact the impedance matching at the water-PDMS interface, which affected the threshold for ADV. The threshold was lowest for mixing ratios of 5:1 and 20:1, and greatest for 9:1. Final bubble volumes were compared with a computational model by Ye & Bull and were found to be consistent. This work is supported by NIH grant R01EB006476.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GS Geophysical: Oceanographic II 200G

8:00AM GS.00001 Double Trouble: Internal Tide Attractors in Double Ridge Systems[1] , PAULA ECHEVERRI, MIT, TITE YOKOSSI, Ecole des Mines, NEIL BALMFORTH, UBC, THOMAS PEACOCK, MIT — A theoretical and experimental study is presented of the generation of internal tides by barotropic tidal flow over topography in the shape of a double ridge. A one-dimensional map is constructed that allows one to track the ray paths of waves reflecting between the ocean surface and topography, and this device is used to expedite the search for internal tide attractors between the ridges, these being attracting, closed ray paths. Calculations are then presented for the steady state scattering of internal tides from the barotropic tide. When attractors are present, these computations break down unless dissipation is also incorporated into the problem, in which case there is significantly enhanced energy conversion in the presence of attractors. We conclude with a direct comparison between theoretical predictions and the results of a laboratory experiment, as well as possible applications to geophysical locations.

[1]This work is supported by the ONR and NSF.

8:13AM GS.00002 Low-mode internal tide generation by topography: an experimental and numerical investigation[1] , MORRIS FLYNN, Univ. of Alberta, PAULA ECHEVERRI, TOM PEACOCK, Mass. Inst. of Technology (MIT), KRAIG WINTERS, Scripps Inst. of Oceanography - UCSD — We summarize recently published work (J. Fluid Mech.) and analyze the low-mode structure of internal tides generated in laboratory experiments and numerical simulations by a two- dimensional ridge in a channel of finite depth. The height of the ridge is approximately half of the channel depth and the regimes considered span sub- to super-critical topography. For small tidal excursions, on the order of 1% of the topographic width, our results agree well with linear theory. For larger tidal excursions, up to 15% of the topographic width, we find that the scaled mode one conversion rate decreases by less than 15%, in spite of nonlinear phenomena that break-down the familiar wave-beam structure and generate harmonics and inter- harmonics. Modes two and three, however, are more strongly affected. For this topographic configuration most of the linear baroclinic energy flux is associated with the mode-1 tide, so our experiments reveal that nonlinear behavior does not significantly affect the barotropic to baroclinic energy conversion in this regime, which is relevant to large scale ocean ridges. This may not be the case, however, for smaller scale ridges that generate a response dominated by higher modes.

[1]Funding provided by NSF and NSERC.

8:26AM GS.00003 A numerical study of oscillatory two-layer stratified flow over three-dimensional topography , LAURA BRANDT, JAMES ROTTMAN, Science Applications International Corporation — A fully-nonlinear numerical model of two-layer stratified flow over three-dimensional topography is used to investigate the generation and propagation of interfacial waves by steady as well as oscillatory flows. Quantitative comparisons of the simulation results are made with shallow-water and weakly nonlinear theories. Qualitative comparisons are made with laboratory experiments and ocean observations.

8:39AM GS.00004 Tidal flow over 3D topography generates out-of-forcing plane harmonics , BENJAMIN KING, HEPENG ZHANG, HARRY L. SWINNEY, University of Texas at Austin — About 1 TW of mixing energy in the ocean comes from internal waves generated by tidal flow over bottom topography [1]. The generation of these waves in three dimensions (3D) remains poorly understood. We use a 3D axisymmetric Gaussian mountain as a model topographic feature and obtain numerical and experimental results for the internal wave field generated by tidal flow. The experiments use a model mountain in a 200 L tank, and particle image velocimetry for imaging. The numerical methods are the same as those in [2], and utilize a finite volume scheme to evaluate the 3D internal wave field. The stratification and forcing frequency are chosen such that $2\omega < N$ (N is the buoyancy frequency), allowing the propagation of second harmonics. Surprisingly, when the maximum topographic slope exceeds the slope of second harmonic wave propagation, strong second harmonics are generated in the direction perpendicular to the tidal forcing direction. At high forcing amplitude, these harmonic waves have higher amplitude than the in-forcing-plane harmonics.

[1] W. Munk and C. Wunsch, Deep-Sea Res. I **45**, 1977 (1998)
[2] B. King, H. P. Zhang, and H. L. Swinney. Submitted to Phys. Fluids

8:52AM GS.00005 The phase lead of shear stress in shallow-water flow over a perturbed bottom , PAOLO LUCHINI, DIM, Universita di Salerno, 84084 Fisciano (SA), Italia, FRANCOIS CHARRU, IMFT, Universite de Toulouse, France — Analysis of the flow over a slowly perturbed bottom (when perturbations have a typical length scale much larger than water height) is often based on the shallow-water (or Saint-Venant) equations, with the addition of a wall-friction term which is a local function of the mean velocity. By this choice small sinusoidal disturbances of wall stress and mean velocity are bound to be in phase with each other. In contrast, studies of shorter-scale disturbances have long established that a phase lead develops between wall stress and mean velocity, with a crucial destabilizing effect on sediment transport over an erodible bed. Our purpose here is to calculate the wall-stress phase lead under large-length-scale conditions, using asymptotic matching techniques for turbulent flow. This calculation provides significant corrections to the shallow-water model.

9:05AM GS.00006 Hydrodynamic instability and rip current generation[1] , JIE YU, ALI MARJANI, North Carolina State University — Rip currents are jet-like offshore flows which are part of the horizontal cell circulations originating inside the surf zone. It is generally acknowledged that alongshore variations in the wave field are essential to rip current generation, however, such a variability can arise from a variety of processes. We present here a linear instability analysis and show that the coupling of waves and evolving currents can lead to a positive feedback, generating rip currents in a system initially alongshore uniform. Preliminary results based on a simplified beach profile show that circulations with alongshore spacing of a few hundreds meters can be initiated by the instability on beaches of typical water depth. Qualitative agreements with observations of natural rip currents are obtained. Extension to complex beach bathymetry is made, and some results are discussed.

[1]This work is supported by National Science Foundation (Grant CBET 0756271).

9:18AM GS.00007 Resonant forcing of nonlinear internal waves by a density current released into a two-layer fluid , BRIAN WHITE, University of North Carolina at Chapel Hill, Marine Sciences, KARL HELFRICH, Woods Hole Oceanographic Institution, Physical Oceanography — The propagation of a density current released by a dam break into a fluid with two-layer stratification is a geophysical fluid dynamics problem relevant to the ocean and atmosphere. Analogous to topographic forcing, a transcritical resonance is observed when the speed of the density current falls within a range, $c_l < c < c_u$, near the linear longwave speed for the two-layer ambient stratification. Within this resonant band, nonlinear solitary waves can be radiated upstream of the front. If the speed of the current is increased beyond this range, by increasing either the height or density of the released fluid, an upstream bore can be generated and is connected to the current front by a rarefaction. This dissipationless bore, also called the conjugate flow, corresponds to the two-layer nonlinear solitary wave solution in the large-amplitude limit, and has speed c_{cs}. As the current speed is increased beyond c_{cs}, steady supercritical flow over the front is observed. The onset of resonance shows qualitative similarities to topographic forcing in the weakly nonlinear limit. However, the upper limit is well-predicted by fully nonlinear theory, with $c_u = c_{cs}$. Unlike the topographic generation problem, the current speed cannot be varied independently, but is determined by the dam height, density, and ambient stratification.

9:31AM GS.00008 Sediment wave formation by unstable internal waves in a turbidity current boundary layer , LUTZ LESSHAFFT, CNRS / Ecole Polytechnique, BRENDON HALL, ECKART MEIBURG, UC Santa Barbara, BEN KNELLER, University of Aberdeen — The bedform of sediment that is deposited from turbidity currents onto the ocean floor is often found to exhibit long-wavelength variations, with crest lines perpendicular to the flow direction ("sediment waves"). A temporal stability analysis, based on the 2D Navier–Stokes equations, reveals the presence of unstable internal waves in the bottom boundary layer of a turbidity current. Instability arises from the interaction between the current and the sediment bed, via the competing effects of particle deposition and erosion. Due to the velocity and density variations within the boundary layer, near-stationary internal waves near the bottom may exist under both sub- and supercritical outer flow conditions. Unstable internal waves display long wavelengths and are typically found to slowly travel upstream. Both features are in qualitative agreement with field observations on sediment waves.

9:44AM GS.00009 Critical Richardson Numbers in Breaking Internal Waves , CARY TROY, Purdue University, ERIN HULT, JEFFREY KOSEFF, Stanford University — The breaking of internal waves is known to be responsible for much of the vertical mixing observed in the ocean, large lakes, and the atmosphere. In order to both correctly model and correctly infer the mixing associated with breaking internal waves, a thorough understanding of the instability mechanism driving these turbulent events is crucial. In this study, a primary indicator of turbulence in stratified flows, the gradient Richardson number (Ri), is examined for internal waves on the verge of instability. We use simultaneous high-resolution scalar (Planar Laser-Induced Fluorescence, PLIF) and velocity (Digital Particle Image Velocimetry, DPIV) measurement techniques to infer the gradient Richardson number of breaking and near-breaking progressive internal waves in a laboratory channel. The results show important deviations from the oft-assumed canonical stability limit of Ri=1/4, which we attribute to the unsteadiness of the internal wave-generated shear driving the instability. Results are compared to inviscid theory based on the normal modes equation. These results have important implications for the diagnosis of turbulent mixing in stratified environments.

9:57AM GS.00010 Dissipation Mechanisms of Internal Solitary-like Waves in the Ocean , KEVIN LAMB, University of Waterloo — Internal solitary-like waves (ISWs) are ubiquitous, highly energetic features in the coastal ocean where they are predominately generated by tide-topography interaction. There are many unanswered questions about the generation and fate of these waves and a better understanding of these processes is necessary for developing parameterizations of their effects for use in large scale models. Several sets of observations have suggested that the mixing associated with ISW trains is important for setting the stratification in some regions of the coastal ocean (e.g, the Scotian Shelf and the Portuguese Shelf). This talk will begin with a discussion of the energetics of large amplitude internal waves. I will then discuss three dissipation mechanisms for ISWs and consequences for mixing: instabilities in the bottom boundary layer, the breaking of shoaling waves, and shear instabilities in the pycnocline. Results from 2D numerical simulations will be presented for all three mechanisms, with a focus on shear instabilities. 3D simulations of shear instabilities have recently been initiated. It is hoped that results from these simulations will also be presented.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GT Vortex Dynamics and Vortex Flows IV 200H

8:00AM GT.00001 Trajectory of an Elliptic Vortex Ring , DEEPAK ADHIKARI, TEE TAI LIM, National University of Singapore — An elliptic vortex ring is known to be unstable due to its oscillatory deformation while propagating. It oscillates periodically at low aspect ratios, but deforms and breaks up into two smaller rings at high aspect ratios. Although studies on elliptic vortex rings have been conducted before, certain aspects of the vortex ring behavior remain unclear; in particular the influence of Reynolds number on their trajectories. Moreover, most of the earlier experimental studies were conducted using flow visualization techniques, which provide only qualitative description on the motion of elliptic vortex rings and not their velocity and vorticity fields. In the present investigation, we focus our attention on the vorticity field during various stages of the vortex ring deformation; particularly the effect of Reynolds number and aspect ratio on the vortex ring trajectory. Experiments are conducted in a water tank using elliptic nozzles of aspect ratios 1, 2 and 3. Obviously, the nozzle aspect ratio of 1 represents a circular nozzle, and the results are included here for comparison. Preliminary results show that the trajectory of elliptic vortex ring of aspect ratio 2 is insensitive to changes in the Reynolds numbers, but this is not the case with the aspect ratio 3, where noticeable deviation of the trajectory at lower Reynolds numbers is observed. The cause of this deviation and its implication will be discussed.

8:13AM GT.00002 Vortex ring impacting on wall , JING LOU, MING CHENG, Institute of High Performance Computing, LI-SHI LUO, Old Dominion University, IHPC TEAM, LI-SHI LUO COLLABORATION — Three dimensional vortex ring impacting a wall at different angles of incidence has been numerically investigated using the lattice Boltzmann model. The detailed flow behavior, vortex evolution, and pressure distribution on wall have been studied systematically with the Reynolds number of $100 < \mathrm{Re} < 1000$, and the impact angle of the range of $0^\circ < \theta < 60^\circ$. Our results show that only when $\mathrm{Re} > 100$, the interaction of the vortex ring with the wall can generates the secondary vortex rings. The evolution of vortex structure is also strongly influenced by θ. The increase of θ will cause a wrap process of the secondary vortex ring and an obvious suppression of the tertiary vortex ring generation. Further more, the study identified new features of vortex structure and its interaction with wall, in particular, for the oblique impact scenarios. A simple model is adopted to describe the basic characteristics of pressure distribution on the wall along the symmetry vortex ring plane at low Reynolds number.

8:26AM GT.00003 Comparison of DNS Determination of the Dynamics of Vortex Rings in Viscous Fluids and Experiment , RUSSELL DONNELLY, ROBERT HERSHBERGER, University of Oregon — We have been studying vortex rings in water for some time [1] and recently became aware of an important paper studying vortex rings by direct numerical simulation (DNS) from Coleman's group at Southampton [2]. There is clearly much to be learned from a comparison of the results in [1] and [2]. The first insight is a comparison of slowing vortex rings, where we find quite similar decay rates at comparable Reynolds numbers. A second insight is gained by noting that they find a time t* needs to elapse before the core adjusts to its vorticity distribution. We find photographically that the ring needs to propagate at least one gun diameter before it adjusts its vorticity. A third insight is that the rings in Fig. 5(b) of [2] do not change much in radius, consistent with the results in Table 2 of our paper [1]. The talk will cover more recent comparisons of the two works including observations of the growth of vortex waves.

[1] I. S. Sullivan, J. J. Niemela, R. Hershberger, D. Bolster and R. J. Donnelly, J. Fluid Mech. 609 319 (2008).
[2] P. J. Archer, T. G. Thomas and G. N. Coleman, J. Fluid Mech.598 201 (2008).

8:39AM GT.00004 Interaction of the 2D vortex patch with the wall. Eruption of the boundary layer phenomenon. , ZIEMOWIT MALECHA, Wroclaw Univ of Technology — The boundary layer eruption phenomenon caused by a 2D patch of vorticity moving above a wall was investigated. It was shown that eruption phenomenon depends on the viscosity (or Reynolds number, Re) of the fluid. There exists a threshold value of Re above which the eruption takes place. The initiation of the eruption goes through the creation of a small recirculation zone near the solid wall. For small Re numbers it disappears but for larger it is strongly stretched in the direction perpendicular to the wall. The terminal state is appearance of a saddle point on streamlines inside the recirculation zone. Next this zone is torn off and portion of the fluid particles from the near wall region are abruptly ejected into the other flow. Further increase of the Reynolds number causes more complex flow. One can observe that eruption is regenerative and that the vortex patch can produce a cascade of secondary vortices. The vortex-in-cell method was employed to investigate the eruption phenomenon.

8:52AM GT.00005 Local Stability Analysis of Fat Vortex Rings , YUJI HATTORI, Tohoku University, KEISUKE HIJIYA, Kyushu Institute of Technology — The stability of fat vortex rings is studied by the geometrical optics method. It is found that Hill's vortex is subject not only to the elliptical instability but also to the curvature instability, which is due to the curvature of vortex tubes and first found for the vortex ring with thin core. A new type of instability is also found; it is a coupled mode of the elliptical and curvature instabilities. The strongest instability is the elliptical instability for a wide area of the vortex, while the coupled instability surpasses the elliptical instability near the surface. The effects of swirl on the instability are investigated. The maximal growth rate becomes small as the magnitude of swirl becomes large.

9:05AM GT.00006 DPIV Measurements of Vortex Ring Interaction with Multiple Permeable Screens[1] , MUSTAFA N. MUSTA, PAUL S. KRUEGER, SMU — Flow visualization of the interaction of a vortex ring impinging on several parallel, transparent permeable screens was made previously for screens with 84% open area ratio. The results indicated the vortex ring split into smaller vortical structures after its interaction with the first screen and exhibited a continuous break down into increasingly irregular flow after interaction with subsequent screens. The flow did not reorganize into a transmitted vortex ring as was observed with vortex rings impinging on a single permeable screen. The present work seeks to provide a more quantitative assessment of the flow through screens using DPIV. DPIV measurements were made using an aqueous solution that was refractive index matched to the transparent screens. Measurements were made for vortex rings interacting with screens with variable spacing and open area ratios of 58%-84%. The vortex rings were generated with a piston-cylinder vortex ring generator using piston stroke-to-diameter ratios of 2-4 and jet Reynolds numbers of 1000-2000. Preliminary results show splitting and decay of the flow vorticity in agreement with the flow visualization.

[1]Support by NSF grant CBET - 0652046 is gratefully acknowledged.

9:18AM GT.00007 Stability of relative equilibria of three vortices[1] , HASSAN AREF[2], Technical University of Denmark — Three point vortices on the unbounded plane have relative equilibria wherein the vortices either form an equilateral triangle or are collinear. While the stability analysis of the equilateral triangle configurations is straightforward, that of the collinear relative equilibria is considerably more involved. The only comprehensive analysis available in the literature, by Tavantzis & Ting [*Phys. Fluids*, **31**, 1392 (1988)], is not easy to follow nor is it very physically intuitive. The symmetry between the three vortices is lost in this analysis. A different analysis is given based on explicit formulae for the three eigenvalues determining the stability, including a new formula for the angular velocity of rotation of a collinear relative equilibrium. A graphical representation of the space of vortex circulations is introduced, and the resultants between various polynomials that enter the problem are used. This approach adds considerable transparency to the solution of the stability problem and provides more physical understanding. The main results are summarized in a diagram that gives both the stability or instability of the various collinear relative equilibria and their sense of rotation.

[1]Supported by the Danish National Research Foundation through a Niels Bohr Visiting Professorship
[2]Permanent address: ESM Dept., Virginia Tech

9:31AM GT.00008 Remarks on Continuation of Inviscid Vortex Flows in the Presence of the Kutta Condition , BARTOSZ PROTAS, Department of Mathematics & Statistics, McMaster University, Canada, FEDERICO GALLIZIO, Dipartimento di Ingegneria Aeronautica e Spaziale, Politecnico di Torino, Italy, ANGELO IOLLO, Institut de Mathematiques de Bordeaux, Universite Bordeaux 1, France, LUCA ZANNETTI, Dipartimento di Ingegneria Aeronautica e Spaziale, Politecnico di Torino, Italy — Our investigation concerns solutions of the steady–state Euler equations in two dimensions featuring finite–area regions with constant vorticity embedded in a potential flow. Using elementary methods of the functional analysis we derive precise conditions under which such solutions can be uniquely continued with respect to their parameters, valid also in the presence of the Kutta condition concerning a fixed separation point. Our approach is based on the Implicit Function Theorem and perturbation equations derived using shape–differentiation methods. These theoretical results are illustrated with careful numerical computations carried out using the Steklov–Poincaré method which show the existence of a global manifold of solutions connecting the point vortex and the Prandtl–Batchelor solution, each of which satisfies the Kutta condition.

9:44AM GT.00009 ABSTRACT WITHDRAWN –

9:57AM GT.00010 Reynolds number effects on the dynamics of the turbulent horseshoe vortex: High resolution experiments and numerical simulations , NIKOLAOS APSILIDIS, SAM RABEN, PANAYIOTIS DIPLAS, CLINTON DANCEY, PAVLOS VLACHOS, Virginia Tech, ALI KHOSRONEJAD, FOTIS SOTIROPOULOS, University of Minnesota — Turbulent flows past wall-mounted obstacles are dominated by dynamically rich, slowly evolving coherent structures producing most of the turbulence in the junction region. Numerical simulations [Paik et al., *Phys. of Fluids* 2007] elucidated the large-scale instabilities but important questions still remain unexplored. One such question is with regard to the effect of the Reynolds number on the dynamics of the turbulent horseshoe vortex (THV). We carry out high-resolution laboratory experiments for the flow past a wall mounted cylinder in a laboratory water tunnel for $\mathrm{Re}_D =$ 26000, 48000 and 117000. We employ the Time-Resolved Particle Image Velocimetry technique to resolve the dynamics of the flow at the symmetry plane of the cylinder and analyze the instantaneous velocity fields using the Proper Orthogonal Decomposition technique. The experimental study is integrated with coherent-structure-resolving numerical simulations providing the first comprehensive investigation of Reynolds number effects on the dynamics of the THV.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GU Granular IV 200I

8:00AM GU.00001 Collective dynamics of floaters on a Faraday wave[1] , CEYDA SANLI, DEVARAJ VAN DER MEER, DETLEF LOHSE, Physics of Fluids-University of Twente — The dynamics of particles floating on a standing Faraday wave is studied experimentally. For low particle concentration it was shown [G. Falkovich et al. Nature 435, 1045 (2005)] that non-wetting particles accumulate at the antinodes of the standing wave. This was found to be a single particle effect. Here, we study what happens when the particle concentration is increased: Surprisingly, we observe that the same particles that cluster at the antinodes for low particle concentration move to the nodes for high concentrations. The explanation of this effect lies in the collective, attractive capillary interaction among particles which counteracts the tendency of the particles to move toward the antinodes. The transition between the two regimes is studied as a function of the concentration and is found to exhibit extremely long transients.

[1]This work is part of the research program of FOM, which is financially supported by NWO.

8:13AM GU.00002 Coarsening of Faraday Heaps: Experiment, Simulation, and Theory , DEVARAJ VAN DER MEER, HENK JAN VAN GERNER, University of Twente, The Netherlands, GABRIEL A. CABALLERO-ROBLEDO, CIMAV, Mexico, KO VAN DER WEELE, University of Patras, Greece, MARTIN A. VAN DER HOEF, University of Twente, The Netherlands — When a layer of granular material is vertically shaken, the surface spontaneously breaks up in a landscape of small Faraday heaps that merge into larger ones on an ever increasing timescale. This coarsening process is studied in a linear setup, for which the average lifetime of the transient state with N Faraday heaps is shown to scale as N^{-3}. We describe this process by a set of differential equations for the peak positions; the calculated evolution of the landscape is in excellent agreement with both the experiments and simulations. The same model explains the observational fact that the number of heaps towards the end of the process decreases approximately as $N(t) \propto t^{-1/2}$.

8:26AM GU.00003 Granular compaction under confinement , NATHAN MUEGGENBURG, Lake Forest College — We report on experiments that explore the impact of confinement on a compacting granular system. When a granular pack is subjected to successive vertical vibrations it undergoes a slow compaction process as individual grains rearrange and pack more closely together. We control the position and applied force of a confining boundary at the top surface of the granular system during these vibrations. This confinement limits the amount of dilation that occurs during the vibrations and significantly reduces the rate of compaction in comparison to the same system with a free top surface.

8:39AM GU.00004 Shocks and Patterns in Continuum Simulations of Oscillated Granular Layers , J. BOUGIE, K. DUCKERT, Physics Department, Loyola University Chicago — We study interactions between shocks and standing wave patterns in continuum simulations of vertically oscillated granular layers. Layers of grains atop a plate with sinusoidal oscillations in the vertical direction leave the plate at some time during the cycle if the accelerational amplitude of oscillation is greater than the acceleration of gravity. Above a critical acceleration, standing waves form stripe patterns. In these same shaken layers, shocks are produced when layers collide with the plate after leaving the plate earlier in the cycle. We simulate vertically shaken layers using numerical solutions of continuum equations to Navier-Stokes order to find number density, average velocity, and granular temperature as functions of time and location within the cell. We compare shocks and standing waves coexisting in this system; pressure gradients produced by shocks play a significant role in the formation of standing wave patterns.

8:52AM GU.00005 Measurement of Density Fluctuations in a Vertically Oscillated Granular Bed at the Onset of Vibrofluidization , JAMES GILCHRIST, KENNETH FORD, COLIN ARMSTRONG, RICHARD EVANS, HUGO CARAM, Lehigh University — The transition from solid- to liquid-like behavior in vertically oscillated granular media is probed through measurements of density fluctuations near the point of vibrofluidization. The intracycle dynamics are used to define the critical vibration acceleration required for vibrofluidization in deep beds. Clear successive shock waves are formed, and the resulting density fluctuations initiated near the free surface propagate downward with increasing energy. When "heating" the bed by vibration, "melting" begins at the free surface.

9:05AM GU.00006 Comparison of two quadrature-based moment methods for simulating dilute granular gases , RODNEY FOX, ALBERTO PASSALACQUA, Iowa State University, PRAKASH VEDULA, University of Oklahoma, JASON YUAN, Iowa State University — A dilute non-isothermal inelastic granular gas between two stationary Maxwellian walls is studied by means of numerical simulations of the Boltzmann kinetic equation with a hard-sphere collision kernel for mono-dispersed particles. Two types of quadrature-based moment methods with different orders of accuracy in terms of the moments of the distribution function are used with four different inelastic collision models. The models differ in the manner with which the moment equations are closed and in the number of moments that can be controlled for a given number of quadrature points. Results from the kinetic models are compared with the predictions of molecular dynamics simulations of a nearly equivalent system.

9:18AM GU.00007 Experimental measurement of the stress tensor in a granular gas , GREG VOTH, WAN JUN YANG, JONAS MISHARA-BLOMBERGER, Wesleyan University, MARK SHATTUCK, City College of New York — We study a quasi-2D granular gas that is vertically vibrated. Precision particle tracking from video at 60 kHz allows us to accurately measure the momentum transfer from individual collisions as well as from particle motion. This allows experimental measurement of the stress tensor. The time averaged stress shows good agreement with the requirement of hydrostatic balance, indicating that we are adequately resolving the stress. Time resolved measurements of the collisional stress show the serrated structure that appears in the shock waves in this system. These measurements allow direct evaluation of the constitutive equation for stress used in hydrodynamic models.

9:31AM GU.00008 A quadrature-based kinetic model for a dilute non-isothermal granular gas[1] , ALBERTO PASSALACQUA, Iowa State University, JANINE GALVIN, US-DOE NETL, PRAKASH VEDULA, University of Oklahoma, CHRISTINE HRENYA, University of Colorado, RODNEY FOX, Iowa State University — A dilute non-isothermal inelastic granular gas between two stationary Maxwellian walls is studied by means of numerical simulations of the Boltzmann kinetic equation with hard-sphere collisions. The behavior of a granular gas in these conditions is influenced by the thickness of the wall Knudsen layer: if its thickness is not negligible, the traditional description based on the Navier-Stokes-Fourier equations is invalid, and it is necessary to account for the presence of rarefaction effects using high-order solutions of the Boltzmann equation. The system is described by solving the full Boltzmann equation using a quadrature-based moment method (QMOM), with different orders of accuracy in terms of the moments of the distribution function, considering moments up to the seventh order. Four different inelastic collision models (BGK, ES-BGK, Maxwell hard-sphere, Boltzmann hard-sphere) are employed. QMOM results are compared with the predictions of molecular dynamics (MD) simulations of a nearly equivalent system with finite-size particles, showing the agreement of constitutive quantities such as heat flux and stress tensor.

[1]This work was supported by US-DOE grant DE-FC26- 07NT43098.

9:44AM GU.00009 ABSTRACT WITHDRAWN –

9:57AM GU.00010 The effect of finite container size on granular jet formation , SYLVAIN JOUBAUD, STEFAN VON KANN, GABRIEL CABALLERO-ROBLEDO, DEVARAJ VAN DER MEER, DETLEF LOHSE, PHYSICS OF FLUIDS TEAM — When an object is dropped into a bed of fine, loosely packed sand, a surprisingly energetic jet shoots out of the bed. In this work we study the effect that boundaries have on the granular jet formation. We did this by (i) decreasing the depth of the sand bed and (ii) reducing the container diameter to only a few ball diameters. These confinements change the behavior of the ball inside the bed, the void collapse, and the resulting jet height and shape. From these results we propose a new explanation for the thick-thin structure observed experimentally and reported previously.

Monday, November 23, 2009 8:00AM - 10:10AM –

Session GV Swimming II 205A-D

8:00AM GV.00001 Repeatability of arm pull patterns in front crawl swimming , LESTER K. SU, JOHN C. KEGELMAN, Johns Hopkins University — The arm pull in human swimming has seen extensive study, particularly involving the front crawl stroke. This work has primarily been aimed either at clarifying the mechanisms of thrust generation by the arm and hand, or at comparing the relative performance of different canonical pulling patterns. In this work we investigate the degree to which swimmers adjust their arm and hand trajectories in response to instantaneous ambient conditions. Video imaging data from competitive swimmers indicates that there may be wide stroke-to-stroke variations in pull trajectories. This suggests that optimal stroking form may be less about a swimmer's ability to repeat idealized pull patterns, than about the swimmer's ability to respond to local flow conditions, or what is referred to in the swimming vernacular as the "feel" for the water.

8:13AM GV.00002 Dynamics of wake structure in clapping propulsion , DAEGYOUM KIM, MORTEZA GHARIB, California Institute of Technology — Some animals such as insects and frogs use a pair of symmetric flaps for locomotion. In some cases, these flappers operate in close proximity or even touch each other. In order to understand the underlying physics of these kinds of motion, we have studied the wake structures induced by clapping and their associated thrust performance. A simple mechanical model with two acrylic plates was used to simulate the power stroke of the clapping motion and three-dimensional flow fields were obtained using defocusing digital particle image velocimetry. Our studies show that the process of vortex connection plays a critical role in forming a downstream closed vortex loop. Under some kinematic conditions, this vortex loop changes its shape dynamically, which is analogous to the process of an elliptical vortex ring switching its minor and major axis. As the length of the plate along the rotating shaft decreases to change an aspect ratio, the downstream motion of the vortex is retarded due to the outward motion of side edge vortices and less propulsive force is generated per the surface area of the plate. The impact of compliance and stroke angle of the plate on wake structures and thrust magnitudes are also presented.

8:26AM GV.00003 Can drag and thrust be separated in undulatory swimming? , NEELESH PATANKAR, Department of Mechanical Engg., Northwestern University, ANUP SHIRGAONKAR, Department of Mechanical Engg., MIT, MALCOLM MACIVER, Department of Mechanical Engg., Northwestern University — Aquatic organisms are motivating new biomimetic underwater vehicles. To that end it is essential to obtain the swimming velocity and efficiency of organisms using reduced order models. The swimming velocity is often determined by equating the drag and thrust on swimming bodies. This has led to many conflicting results in the past. It has been proposed that one of the root causes of the disagreements is that, in general, drag and thrust on swimming bodies can not be separated from each other. This is considered to be true when movement is generated by undulations as in anguilliform, gymnotiform, and balistiform modes of swimming, among others. We did high-resolution numerical simulations to study the forces acting on the undulatory ribbon fin of a gymnotiform swimmer – the black ghost knifefish. In spite of the above expectations, we have surprisingly found a new way to approximately decompose the net force into drag and thrust producing mechanisms in undulatory swimming modes. Such decomposition is unexpected for non-linear finite Reynolds number problems. This result appears conceptually analogous to how a linearization of the Navier–Stokes equations, or Carrier's equation, captures the drag-determining features of the flow around objects.

8:39AM GV.00004 The effects of fluid viscosity on undulating swimmers , JOSUE SZNITMAN, XIAONING SHEN, PAULO ARRATIA, University of Pennsylvania — The swimming behavior of the nematode *C. elegans* ($L \approx 1$ mm) as a function of the surrounding fluid viscosity μ is investigated using both particle- and nematode-tracking methods. Nematode tracking data show that *C. elegans* move in a highly periodic fashion characterized by traveling waves. The nematode swimming speed U decays nonlinearly with increasing fluid viscosity such that $U \sim \mu^{-0.2}$. Velocimetry data shows flow re- circulation regions along the nematode's body. The velocity profiles measured in the direction normal to the swimming nematode show a decay that is similar for fluid viscosities ranging from from 1 cP to 20 cP. The normalized velocity decays follow a single mater curve with d/L as the independent variable, where d is the normal distance from the swimming nematode. This result suggests that *C. elegans* may be a good canditate to investigate low Re locomotion.

8:52AM GV.00005 High speed x-ray observation of a sand swimming lizard[1] , DANIEL GOLDMAN, School of Physics, Georgia Tech, RYAN MALADEN, Bioengineering Program, Georgia Tech, YANG DING, School of Physics, Georgia Tech — We use high-speed x-ray imaging to reveal how a small (10 cm) desert dwelling lizard, the sandfish (*Scincus scincus*), swims within a granular medium, and how its locomotion is affected by the volume fraction ϕ of the media [2]. We use an air fluidized bed to prepare 0.3 mm glass beads (similar in size to desert sand) into naturally occurring loose ($\phi = 0.58$) and close ($\phi = 0.62$) packed states. On the surface, the lizard uses a standard diagonal gait, but once below the surface, the organism no longer uses limbs for propulsion. Instead it propagates a large amplitude single period sinusoidal traveling wave down its body and tail to propel itself at speeds up to ≈ 1 body-length/sec. For fixed ϕ the animal increases forward swimming speed v_f by increasing temporal frequency f. For fixed f, v_f is independent of ϕ, despite resistance forces that nearly double from loose to close packed states. Surprisingly, the greatest sandfish velocity (and f) occur in the close packed state.

[1]Support from NSF Physics of Living Systems and Burroughs Wellcome Fund
[2]Maladen et. al, Science, **325**, 314, 2009

9:05AM GV.00006 Resistive force theory for sand swimming , YANG DING, School of Physics, Georgia Tech, RYAN MALADEN, Bioengineering Program, Georgia Tech, CHEN LI, DANIEL GOLDMAN, School of Physics, Georgia Tech — We discuss a resistive force theory [1] that predicts the ratio of forward speed to wave speed (wave efficiency, η) of the sandfish lizard as it swims in granular media of varying volume fraction ϕ using a sinusoidal traveling wave body motion. In experiment $\eta \approx 0.5$ independent of ϕ and is intermediate between $\eta \approx 0.2$ for low Re Newtonian fluid undulatory swimmers like nematodes and $\eta \approx 0.9$ for undulatory locomotion on a deformable surface. To predict η in granular media, we developed a resistive force model which balances thrust and drag force over the animal profile. We approximate the drag forces by measuring the force on a cylinder (a "segment" of the sandfish) oriented at different angles relative to the displacement direction. The model correctly predicts that η is independent of ϕ because the ratio of thrust to drag is independent of ϕ. The thrust component of the drag force is relatively larger in granular media than in low Re fluids, which explains why η in frictional granular media is greater than in viscous fluids.

[1]Maladen et. al, Science, **325**, 314, 2009

9:18AM GV.00007 Sandfish numerical model reveals optimal swimming in sand , RYAN MALADEN, Bioengineering Program, Georgia Tech, YANG DING, ADAM KAMOR, School of Physics, Georgia Tech, ANDREW SLATTON, Computer Science, Ohio State University, DANIEL GOLDMAN, School of Physics, Bioengineering Program, Georgia Tech — Motivated by experiment and theory examining the undulatory swimming of the sandfish lizard within granular media [1], we study a numerical model of the sandfish as it swims within a validated soft sphere Molecular Dynamics granular media simulation. We hypothesize that features of its morphology and undulatory kinematics, and the granular media contribute to effective sand swimming. Our results agree with a resistive force model of the sandfish and show that speed and transport cost are optimized at a ratio of wave amplitude to wavelength of ≈ 0.2, irrespective of media properties and preparation. At this ratio, the entry of the animal into the media is fastest at an angle of $\approx 20^\circ$, close to the angle of repose. We also find that the sandfish cross-sectional body shape reduces motion induced buoyancy within the granular media and that wave efficiency is sensitive to body-particle friction but independent of particle-particle friction.

[1]Maladen et. al, Science, **325**, 314, 2009

9:31AM GV.00008 No slip locomotion of hatchling sea turtles on granular media , NICOLE MAZOUCHOVA, School of Biology/ Georgia Tech, CHEN LI, School of Physics/Georgia Tech, NICK GRAVISH, School of Physics/ Georgia Tech, ANDREI SAVU, School of Physics/Georgia Tech, DANIEL GOLDMAN, School of Physics/ Georgia Tech — Sea turtle locomotion occurs predominantly in aquatic environments. However after hatching from a nest on a beach, the juvenile turtles (hatchlings), must run across several hundred meters of granular media to reach the water. To discover how these organisms use aquatically adapted limbs for effective locomotion on sand, we use high speed infrared video to record hatchling Loggerhead sea turtles (*Caretta caretta*) kinematics in a field site on Jekyll Island, GA, USA. A portable fluidized bed trackway allows variation of the properties of the granular bed including volume fraction and angle up to the angle of repose. Despite being adapted for life in water, on all treatments the turtles use strategies similar to terrestrial organisms when moving on sand. Speeds up to 3 BL/sec are generated not by paddling in sand, but by limb movement that minimizes slip of the flippers, thus maintaining force below the yield stress of the medium. We predict turtle speed using a model which incorporates the yield stress of the granular medium as a function of surface angle.

9:44AM GV.00009 Tree-inspired Piezoelectric Energy Harvesting , WILLIAM HOBBS, Georgia Institute of Technology, DAVID HU — We design and build a tabletop wind energy harvester inspired by the swaying of trees. The device consists of cantilevered cylinders ("tree trunks") arranged linearly downwind. The bases of the cylinders contain piezoelectric transducers that capture energy from vibration of the cylinder transverse to the flow. For a particular Reynolds number, and ratio of vortex shedding frequency to cylinder natural frequency, we experimentally measure the power generated ($\sim$ 1 micro-watt) as a function of cylinder arrangement. We report optimal spacings for generating peak power. We also report the distribution of power down the array. We qualitatively account for these trends using flow visualizations of vortex shedding using a flowing soap film dynamically matched with our piezoelectric system.

9:57AM GV.00010 Flow energy harvesting – another application of the biomimetic flapping foils , QIANG ZHU, ZHANGLI PENG, UC San Diego — Imitating fish fins and insect wings, flapping foils are usually used for biomimetic propulsion. Theoretical studies and experiments have demonstrated that through specific combinations of heaving and pitching motions, these foils can also extract energy from incoming wind or current. Compared with conventional flow energy harvesting devices based upon rotating turbines, this novel design promises mitigated impact upon the environment. To achieve the required motions, existing studies focus on hydrodynamic mode coupling, in which a periodic pitching motion is activated and a heaving motion is then generated by the oscillating lifting force. Energy extraction is achieved through a damper in the heaving direction (representing the generator). This design involves a complicated control and activation system. In addition, there is always the possibility that the energy required to activate the system exceeds the energy recovered by the generator. We have discovered that a much simpler device without activation, a 2DOF foil mounted on a rotational spring and a damper undergoing flow-induced motions can achieve stable flow energy harvesting. Using Navier-Stokes simulations we predicted different behaviors of the system during flow-induced vibrations and identified the specific requirements to achieve controllable periodic motions essential for stable energy harvesting. The energy harvesting capacity and efficiency were also determined.

Monday, November 23, 2009 8:00AM - 10:10AM –
Session GW Mini-Symposium on Fluid Dynamics at Super-repellent Surfaces 208A-D

8:00AM GW.00001 Unusual dynamical properties of water repellent materials , DAVID QUÉRÉ[1], ESPCI and Ecole polytechnique, Paris, France — The reason why water repellent materials have been attracting such an attention for about ten years is mainly related to the remarkable dynamical properties they generate. Paradoxically, apart from the question of slip on such surfaces, only a few quantitative studies were devoted to these properties. In our talk, we plan to describe different ways of reaching superhydrophobicity, by texturing the underlying solid or the deposited liquid, or by heating the substrate. Then we list and describe the different specific dynamical behaviours which are observed, such as ultra-low hysteresis, (fast) running, bouncing or self-motion

[1]Other contributors to this talk: Pascale Aussillous, Christophe Clanet, Lei Jiang, Marie Le Merrer, Guillaume Lagubeau, Mathilde Reyssat and Denis Richard.

8:26AM GW.00002 Drag reduction in laminar and turbulent flows past superhydrophobic surfaces , JONATHAN P. ROTHSTEIN, University of Massachusetts Amherst — A series of experiments and direct numerical simulations (DNS) will be presented which demonstrate significant drag reduction for both laminar and turbulent flows of water through channels using superhydrophobic surfaces with well-defined micron-sized surface roughness. The surfaces are fabricated from PDMS to incorporate precise patterns of ridges or posts that can support a shear-free air-water interface. A flow cell is used to measure the pressure drop and velocity profile as a function of the flow rate for a series of channel geometries and superhydrophobic surface designs. DNS are performed for flow past superhydrophobic surfaces which both complement and extend the range of geometries and Reynolds number obtained in the experiments. We will show that drag reductions up to 75% and slip lengths up to 150μm can be obtained in turbulent flows past superhydrophobic surfaces. Additionally, we will show that slip along the air water interface forestalls the transition from laminar to turbulent flow. The drag reduction is found to increases with increasing post/ridge spacing and the fraction of air-water interface. In turbulent flows, the drag reduction increases with Reynolds number before eventually reaching a plateau. These results suggest that in turbulent flows, the drag reduction scales with the thickness of the viscous sublayer and not the overall channel height as in laminar flows.

8:52AM GW.00003 Controlling inertia dominated flows with super-repellent surfaces , CHRISTOPHE YBERT, CNRS — The possibility to affect liquid flows through surface properties was naturally put forward by the recent emergence of small-scales fluidic devices, as downsizing invariably emphasizes the role of surfaces, with respect to bulk properties. Such strategy of flow modification by surface effects is *a priori* restricted to the natural scales setting the interactions between the surface and the nearby liquid that is, essentially to nanometric scales. In this context, super-repellent surfaces have emerged as possessing not only remarkable (non-)wetting properties but also unique dynamical properties. The latter manifest on their ability to promote large boundary slippage, characterized by slip lengths from 1 to hundreds of microns, that make them capable of modifying flows up such micro-scales. More fundamentally, this raises the question of how far this strategy of flow control through surfaces can be pushed, and of how deep the modification of liquid flows close to super-repellent surface is: can it persist at large scales or large velocities? After briefly going through the properties of super-repellent surfaces in laminar viscous flows, I will discuss their impact on different macro-scale experimental configurations involving inertia-dominated flows. Focusing on splashing and dripping phenomena - the latter being associated to the well-known teapot effect- I will show that although surface effects are usually ignored in such situations, in view of the large values of the Weber number, it is still possible to shape the liquid flows by tailoring surface properties, with optimized effects obtained for super-repellent surfaces.

9:18AM GW.00004 Promoting Giant Liquid Slip on Omniphobic Surfaces with Re-entrant Textures , GARETH H. MCKINLEY[1], Massachusetts Institute of Technology — It is now well-known that by controlling the surface chemistry and topographic details of a textured surface one can generate composite air-liquid-solid interfaces or "Cassie-Baxter states." If the surface topography becomes *re-entrant* (i.e. multi-valued) and very low surface energy coatings are employed, then it becomes possible to create superoleophobic surfaces that are not wetted even by low-tension liquids such as oils and alcohols. Such robustly liquid-repellent (or *omniphobic*) surfaces lead to very high apparent contact angles, low contact angle hysteresis and the possibility of "giant liquid slip" over the microscopic air pockets or "plastron film" trapped in the re-entrant textured surface. Lithographic fabrication approaches have been proposed for developing such re-entrant textured surfaces - a major challenge with such approaches is to develop viable manufacturing protocols that can be readily extended to larger areas. In the present work we use periodic woven fiber meshes of controlled feature size and weave, coupled with a simple elastomeric fluoropolymer dipcoating protocol, to prepare a series of model re-entrant and friction-reducing surfaces. We use parallel-plate rheometry to explore the degree of friction reduction that can be achieved as the geometric details of the meshes are varied and compare the experimental results with recent scaling theories. Apparent slip lengths of greater than 500μm are observed for optimal textures and coatings. By varying the thickness and viscosity of the sheared fluid layer, the robustness of the plastron air film to increasing pressure differentials can also be explored in parallel.

[1]In collaboration with: Wonjae Choi, Shreerang S. Chhatre, Kenneth (Kyoo Chul) Park, Robert E. Cohen, Massachusetts Institute of Technology

9:44AM GW.00005 Development of Surface Structures for Large Effective Slip: How Much Slip Is Possible in Ideal, Lab and Real Conditions? , CHANG-JIN KIM, University of California, Los Angeles — An ideal condition to reduce the drag of a liquid flowing on a solid surface is maintaining a lubricating gas layer between the solid and the liquid. For water flowing on a 1 or 10 μm-thick air layer, for example, the slip length is calculated to be roughly 50 or 500 μm, respectively - large enough to benefit a wide range of engineering applications. Unfortunately, however, the above ideal water-levitating condition is only imaginary, because such a liquid-gas meniscus cannot be sustained in nature. Instead, water-repelling structured surfaces bring us closer to the imaginary condition by minimizing the liquid-solid interface and keeping the water mostly on a layer of air. The underlying goal in developing a large-slip surface is, therefore, to create a condition as close as possible to the uniform air lubrication, which is often overlooked. For example, while a large contact angle on a superhydrophobic surface helps keep the liquid fakir, note that once levitated, the contact angle has little effect on increasing the slip length. Instead, the geometrical parameters of the surface structures, e.g., air fraction, pitch and depth of the structures, are the determining factors. A series of development efforts to create surfaces that bring us closer to the ideal air-lubricating condition will be presented, with the slip length currently measured as large as 400 μm. However, it will be also noted that they are valid only in laboratory conditions, where the sample is fabricated to near perfection and the pressure in the flowing liquid is under strict control. In real-life engineering conditions, which include high and fluctuating pressure, defective surfaces, and liquids full of impurities and particles, it remains to be seen if we will ever be able to create a slip surface that can be field-deployed - a millennium-old dream.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HA Turbulent Boundary Layers: Roughness Effects 101A

10:30AM HA.00001 Turbulent boundary layer over a small, 2-D, k-type roughness[1] , M.P. SCHULTZ, R.J. VOLINO, K.A. FLACK, U.S. Naval Academy — Previous results (Volino, Schultz & Flack JFM to appear) indicate that 2-D, k-type roughness gives rise to significant changes in the turbulence structure well into the outer layer. This stands in contrast to the similarity that has been observed between flows over smooth walls and 3-D roughness outside the roughness sublayer and seems to indicate that there is a fundamental difference in the response of boundary layers to 2-D and 3-D roughness. In the previous study, however, the relative roughness height was fairly large ($k/\delta = 0.03$; $k_s/\delta = 0.42$) leaving open the possibility that the observed effect was simply due to the strength of the wall perturbation. In the present paper, experimental results are presented for a zero pressure gradient boundary layer over a surface with 2-D, k-type roughness. The roughness consisted of transverse bars of square cross section with a pitch of $8k$ like the previous 2-D rough wall but with k reduced by a factor of 7. The relative roughness height was significantly reduced ($k/\delta = 0.007$; $k_s/\delta = 0.055$) in the present case. The results were qualitatively similar to those over the larger roughness. Specifically, the Reynolds stresses were significantly larger over the 2-D roughness than over a smooth wall, and length scales based on two-point spatial correlations were longer for the 2-D roughness.

[1]This work was supported by ONR.

10:43AM HA.00002 Boundary Layer Measurements over 2/3-D Roughness[1] , KAREN FLACK, MICHAEL SCHULTZ, VOLINO RALPH, United States Naval Academy — Boundary layer flows over three-dimensional roughness show remarkable similarity to smooth-wall flows outside a region near the roughness elements. In contrast, the turbulence statistics for flows over two-dimensional bar roughness have exhibited significant differences in the outer layer compared to smooth-wall flows. It is not clear whether the difference is due to the larger scales imposed on the flow due to the spanwise extent of the roughness, or if the boundary layer is strongly perturbed by, in essence, being repeatedly tripped by the spanwise elements. To address this question, measurements have been made over spanwise rows of cubes with a spanwise spacing of one roughness height and a streamwise spacing of seven roughness heights. Results are compared to bars of the same height and streamwise spacing.

[1]Sponsored by the Office of Naval Research

10:56AM HA.00003 Turbulence Statistics Over 3D Roughness in a Turbulent Channel Flow[1] , JIARONG HONG, JOSEPH KATZ, Johns Hopkins University, MICHAEL SCHULTZ, US Naval Academy — This study focuses on the near-wall flow field within a turbulent channel flow over a rough surface. Performing experiments in a facility containing a fluid with the same refractive index as the acrylic rough plate facilitates PIV measurements very near the wall. Presently, the flow in the vicinity of uniformly distributed 0.45mm high pyramids at Re_τ=3400-5418 is resolved at a vector spacing of 63um, ∼9 wall units. Data in a streamwise-wall-normal plane shows that below one roughness height, there is an upsurge of <u'u'>, and there are substantial spatial variations in <u'u'>, <u'v'> and <v'v'>, which rapidly diminish father from the wall. All Reynolds stress components peak above the forward face of roughness, with <v'v'> peaked slightly downstream of the others. The in-plane turbulent kinetic energy (TKE) production peaks deep in the roughness sublayer, especially near the pyramid crest. Both <u'u'> $\partial U/\partial x$ and -<u'v'> $\partial U/\partial y$ are significant contributors. Measurements in a streamwise-spanwise plane located within the roughness sublayer show spatial variability of <w'w'> and <u'w'>, and their contributions to TKE production.

[1]Sponsored by ONR

11:09AM HA.00004 Flow of Turbulent Boundary Layers Over Low-Order Representations of Irregular Surface Roughness , R. MEJIA-ALVAREZ, K.T. CHRISTENSEN, Univ. of Illinois — The relative impact of various topographical scales present within irregular surface roughness on a turbulent boundary layer is explored. 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 65% 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 acquire ensembles of velocity fields for all cases. Comparison of single-point statistics indicates that a 16-mode model of the full surface generally reproduces the statistical character of flow over the full surface. In the outer region, both the 5- and 16-mode models reproduce the characteristics for flow over the full surface in accordance with Townsend's wall similarity hypothesis. However, both surface representations fail to reproduce important details of the Reynolds-shear-stress-producing events within the roughness sublayer.

11:22AM HA.00005 A Shape-Hessian based analysis of roughness effects on turbulence , SHAN YANG, GEORG STADLER, ROBERT MOSER, UT-Austin, ICES, OMAR GHATTAS, UT-Austin — One of the difficulties with evaluating the effects of roughness on wall-bounded flows is that the commonly used metric for roughness effects, the equivalent sand-grain roughness height, is determined not from the topography of the roughness, but from the measured effect of the roughness on the flow. It would be much more useful if the effects of roughness could be predicted directly from the roughness topography. To do this, we characterize the mapping from roughness topography to fluid dynamics impact (in this case the drag) by examining its shape gradient and shape Hessian. The eigenfunctions and eigenvalues of the shape Hessian are studied as they describe how the fluid dynamics impact changes with the roughness. For flat boundaries, the Fourier modes can be proven to be the eigenfunctions of the shape Hessian. Further, the flat boundary is a stationary point (a minimum) of this mapping and the eigenvalues depend on the wavenumber and the Reynolds number. A priori knowledge of the eigenfunctions allows the entire shape Hessian operator to be determined from a single solution of state, incremental state, adjoint and incremental adjoint equations, making determination of the adjoint feasible, even for turbulent flows. For transient Navier Stokes flow (i.e. turbulence), DNS will be used to find the Hessian in this way. The adjoint equations are solved backwards in time, requiring the complete time history of the state solution. The algorithmic and computational challenges of these calculations are discussed.

11:35AM HA.00006 Continuous Wavelet Analysis of a Highly Irregular Roughness Topography for Turbulence Studies , YANHUA WU, HUIYING REN, Wright State University — The realistic surfaces encountered in engineering wall-bounded turbulent flows have complex roughness topographies and occupy a wide range of various roughness parameters such as length scales, aspect ratios, and orientation angles, etc. In order to quantify the effects of those roughness parameters of a highly irregular rough surface on the turbulent boundary layers, the present study used continuous Mexican hat and Morlet wavelets to extract the dominant aspect ratio, length scale and orientation of the surface's random roughness elements. The roughness under the current study is replicated from a turbine blade damaged by deposition of foreign materials. For this particular roughness topography, the continuous wavelet analysis reveals that the dominant aspect ratio, length scale and orientation angle are 1/5, 24 mm and 4°, respectively.

11:48AM HA.00007 Internal Layer Hierarchy in Rough-Wall Turbulent Boundary[1] , FARAZ MEHDI, CALEB MORRILL-WINTER, RACHEL EBNER, JOSEPH KLEWICKI, University of New Hampshire — The existence of an internal layer hierarchy is centric to the characteristic properties of wall-bounded turbulent flows. Its presence, which is revealed through an analysis of the mean momentum balance (MMB), accounts for the dynamics undergoing a continuous self-similar variation over a length scale range spanning the viscous length scale to the outer scale, $\nu/u_\tau \leq \ell \leq \delta$. Surface roughness introduces multiple new length scales which are often reduced (for simplification and comparison) to a single "working" scale given by the equivalent sandgrain roughness k_s^+. We report on our continuing efforts to study how this imposition modifies the continuous hierarchy of scaling layers admitted by the MMB. The establishment of log-like behavior closer to the wall in rough-wall flows is one such effect. It is speculated to be the direct consequence of the roughness causing the vorticity field to three-dimensionalize more rapidly compared to a smooth-wall. Data sets comprising of experiments being performed at UNH and high quality data sets available in the literature are being used for this combined roughness–Reynolds number study. The current experiments are conducted in a 8m long boundary layer wind-tunnel. Roughness is introduced in the form of sandpaper attached to the entire lower wall and profiles are taken using hot-wires and two-dimensional laser velocimetry.

[1]The support of the ONR (N000140810836, grant monitor Ronald Joslin) is gratefully acknowledged.

12:01PM HA.00008 Perturbation of a Turbulent Boundary Layer by Spatially Impulsive Dynamic Roughness[1] , I. JACOBI, Graduate Aerospace Laboratories, California Institute of Technology, C. GONZALEZ, California Polytechnic University, Pomona, M. GUALA, B.J. MCKEON, Graduate Aerospace Laboratories, California Institute of Technology — The effect of a spatially impulsive patch of dynamic roughness on a zero pressure gradient, turbulent boundary layer is experimentally studied. The roughness patch is mechanically actuated at a range of frequencies on the order of the boundary layer burst frequency.The downstream evolution of the perturbed boundary layer is then measured by hot-wire anemometry and particle image velocimetry. Velocity profiles and spectral characteristics of the dynamic roughness case are compared with those in the spatially impulsive static roughness case, over a range of roughness amplitudes in the inner region of the boundary layer. The impact of the dynamic roughness on the near-wall turbulence in close proximity to the roughness as well as the recovery of the outer layer farther from the impulse are explored. The additional timescale introduced by the dynamic roughness provides a potential tool for the manipulation of the structure of boundary layers and, by extension, flow control.

[1]This study is supported by the Air Force Office of Scientific Research under grant #FA9550-08-1-0049 (program manager John Schmisseur).

12:14PM HA.00009 Relating turbulent friction and energy spectrum in rough-pipe flows , CARLO ZUNIGA ZAMALLOA, PINAKI CHAKRABORTY, NIGEL GOLDENFELD, GUSTAVO GIOIA, University of Illinois at Urbana-Champaign — The classical experiments on turbulent flows over rough walls date back to Nikuradse in 1933. Nikuradse reported measurements of friction factor (f), or non-dimensional wall shear stress, as a function of the Reynolds number (Re) of the flow for pipes of six different values of roughness. A recent theory makes a mathematical link between the turbulent energy spectrum and the functional dependence of f on Re (PRL, 044502, 96, 2006). Here we perform experiments on rough pipes to test this mathematical link. To that end, we measure f vs Re as well as the attendant turbulent energy spectrum. Our experimental results are in good with the mathematical link predicted by the theory.

12:27PM HA.00010 Experimental Investigation of Flow Structures in the Boundary Layer over a Moving Rough Wall , KYUNG-HOON SHIN, JIAN SHENG, U. Minnesota — Understanding turbulent flows over a moving surface has overarching implications in understanding rotary machinery and buffer-layer dynamics. We investigate effects imposed by such a boundary on flow structures and further elucidate its role to near wall dynamics. The wind-tunnel experiments are conducted over a 1.5x0.9m belt moving in the spanwise direction to impose crossflow wall stresses to a flat-plate turbulent boundary layer. Stream-wise roughness stripes (h=1mm) are patterned over the entire surface at the spacing of 1cm to imitate transverse waveform. The belt rotates at 10-50 rpm to generate effective wave speeds of 0.25-1.27 m/s. The free stream velocities are 5-20 m/s resulting in $Re_x = (0.5-2)\text{x}10^6$, and Re_h=*500-2000* based on the roughness height. PIV is used to measure mean velocity profiles and fluctuation fields in three $x-y$ planes located at leading, mid, and trailing edges of the belt. Measurements on two $x-z$ planes located at $y=h$ and $50\delta_v$ are also performed to assess effects of moving roughness on near wall structures. Turbulence statistics, i.e. distributions of Reynolds stress, turbulent kinetic energy, budgets and dissipation, are provided.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HB Turbulence: Fundamentals II 101B

10:30AM HB.00001 Multi-scale geometric analysis of Lagrangian structures in isotropic turbulence[1] , YUE YANG, D.I. PULLIN, IVAN BERMEJO-MORENO, California Institute of Technology — The recently developed multi-scale methodology (see J. Fluid Mech. 603, 101-135, 2008) is applied to study the non-local geometry of finite-sized Lagrangian structures in forced isotropic turbulence. A particle backward-tracking method was first applied to obtain the Lagrangian scalar field ϕ governed by the pure advection equation. The temporal evolution of Lagrangian structures was obtained by extracting iso-surfaces of ϕ with resolution 1024^3. The multi-scale geometric analysis was then applied on the evolution of ϕ to extract structures at different length scales and to characterize their non-local geometry in a "visualization space" of reduced geometrical parameters. We observe an evolutionary breakdown of Lagrangian blobs that first distort and then stretch into sheets. Compared with the statistical geometry of instantaneous passive scalar and enstrophy fields in turbulence, Lagrangian structures tend to exhibit more prevalent sheet-like shapes at inertial-range and small scales. Furthermore, after a finite time, the evolutionary geometry of Lagrangian structures appears to be insensitive to the form of the initially smooth Lagrangian scalar field.

[1]Supported by the NSF.

10:43AM HB.00002 Bridging from Eulerian to Lagrangian statistics in turbulent flows , OLIVER KAMPS, Center for Nonlinear Science, University of Muenster, RUDOLF FRIEDRICH, Institute for Theoretical Physics, University of Muenster, HOLGER HOMANN, RAINER GRAUER, Institut fuer Theoretische Physik I, Ruhr-Universitaet Bochum — The problem of relating Lagrangian and Eulerian statistics is a long standing problem in basic and applied turbulence research. Motivated by the investigation of Lagrangian statistics in the inverse cascade regime of 2D turbulence and in fully developed 3D Turbulence we adress the question of relating Lagrangian and Eulerian velocity increment statistics. It turns out that a formal connection between both frames of reference can be established via transition probabilities, which can be estimated from numerical simulations. We also focus on the validity of Corrsin's independence hypothesis in the context of 2D and 3D flows which is of general interest for turbulent transport and mixing processes.

10:56AM HB.00003 Eulerian statistics from Lagrangian dynamics , FEDERICO TOSCHI, Technische Universiteit Eindhoven, The Netherlands, LUCA BIFERALE, University of Tor Vergata, Roma, Italy, ENRICO CALZAVARINI, ENS-Lyon, France, ANDREA SCAGLIARINI, University of Tor Vergata, Roma, Italy — Some statistical properties of fluid dynamics turbulence are very difficult to study usually because of the extremely low signal to noise ratio. Here we show how the Lagrangian approach can help unravelling elusive statistics of Eulerian turbulence. Studying the collective dynamics of bunches of light particles we are able to measure the time of life of Eulerian vortex filaments (arXiv:0908.0205). By means of Lagrangian tracers we are able to measure the full multi-scale and multi-time correlation functions in 3D turbulence.

11:09AM HB.00004 Lagrangian intermittency and time–correlations in two–dimensional turbulence[1] , KAI SCHNEIDER, M2P2-CNRS & CMI, Aix-Marseille University, Marseille, France, WOUTER BOS, LMFA-CNRS, Ecole Centrale de Lyon, University of Lyon, Ecully, France, BENJAMIN KADOCH, SALAH NEFFAA, M2P2-CNRS & CMI, Aix-Marseille University, Marseille, France — The statistical properties of Lagrangian particle transport are investigated in dissipative drift-wave turbulence considering the Hasegawa-Wakatani model. This model allows to study the change in dynamics for different turbulent flow regimes by varying the adiabaticity parameter c. The hydrodynamic limit is obtained for c = 0, while the geostrophic limit is recovered for c $\gg$ 1. For c of order unity the quasi-adiabatic regime, relevant for fusion plasmas in Tokamaks, is obtained. By means of direct numerical simulation we consider four values for c and show that the Lagrangian dynamics is only intermittent in the hydrodynamic regime, while the other regimes are not. This is illustrated by considering the probability density function (PDF) of velocity increments, autocorrelation functions of velocity and acceleration and structure functions. In both, quasi-adiabatic and quasi-geostrophic regimes the PDFs of acceleration exhibit exponential tails. This behaviour is due to the pressure term in the acceleration and not a signature of intermittency. Furthermore the long time correlation of the modulus of acceleration is found for all regimes which hence does not imply intermittency either.

[1]We thankfully acknowledge financial support from ANR, project M2TFP.

11:22AM HB.00005 Modulated Turbulence , HAKKI ERGUN CEKLI, WILLEM VAN DE WATER, Eindhoven University of Technology — Many turbulent flows are subject to periodic modulation, examples are the pulsatile flow of blood through arteries and geophysical flows driven by periodic tides. When the modulation is slow, the turbulence will adjust adiabatically, but when the modulation period comes close to an internal time scale of the flow, the turbulence may resonate with the driving. The possibility of a resonance is intriguing as one may object that turbulence does not have a single dominant timescale, but a continuum of strongly fluctuating times. In our experiment we periodically modulate a turbulent windtunnel flow with an active grid. An active grid is a regular grid of axes with attached vanes which are rotated by servo motors. By controlling the time-dependent angle of all axes precisely, the grid cycles through a sequence of transparency patterns. Thus we modulate turbulence in space, characterized by these patterns, and time, characterized by the modulation frequency. We consider 3 distinct spatial modes, all share the same transparency sequence. We find a large resonant enhancement of the mean turbulent dissipation rate at a modulation frequency which equals the large-eddy turnover rate. Thus, we find the best frequency to inject energy in a turbulent flow. The resonant enhancement depends on the spatial mode of the grid, but all spatial modes share the same behavior of the response. Modulation only affects the large-scale spatial structure of turbulence, leaving the small-scale motion unaltered.

11:35AM HB.00006 Velocity kinematic relations in a turbulent flow past a grid , ALEX LIBERZON, Tel-Aviv U., ROI GURKA, Ben-Gurion U., GREGORY KOPP, PARTHA SARATHI, U. Western Ontario, ARKADY TSINOBER, Tel-Aviv U. — We present velocity kinematic relations, involving average and difference of the longitudinal velocity component of the two points at distance r: $u_+ = u(x+r) + u(x)$ and $u_- = u(x+r) - u(x)$, obtained using PIV measurements in a turbulent flow of water past a grid. The present study follows recent numerical and experimental studies, that demonstrated analytical and empirical evidence of the relations, their validity and it emphasizes the physical meaning of the relations. The relations that contain both the large (u_+) and small (u_-) scale quantities emphasize the non-local aspects of turbulent flows. For example, the pure kinematic relation of Hosokawa in conjunction with the the Kolmogorov 4/5 law leading to the $\langle u_+^2 u_- \rangle = \langle \epsilon \rangle r/30$ shows that the that the large and small scale quantities are correlated contrary to what is suggested by the commonly used sweeping decorrelation hypothesis. Some relations are purely kinematic and some are dynamic, i.e. involving $\langle \varepsilon \rangle$, like the Kolmogorov 4/5 law. The most important aspect is that pure kinematic relations that emphasize the non-local effects, become dynamically significant. Furthermore, we suggest that many of these relations could be used for validation of experimental results.

11:48AM HB.00007 A Kinetic Description of Turbulent Velocity and Vorticity Distributions , MICHAEL WILCZEK, ANTON DAITCHE, RUDOLF FRIEDRICH, Institute for Theoretical Physics, University of Muenster — The single-point statistics of velocity and vorticity in fully developed homogeneous and isotropic turbulence displays very distinct features. While the former is known to exhibit nearly Gaussian behavior, the latter develops strong non-Gaussian tails, mirroring the differing spatial structure of both fields. We analyze these statistics within the framework of the Lundgren-Monin-Novikov hierarchy, which allows to derive kinetic equations for the evolution of the probability functions from first principles. The unclosed terms are estimated with the help direct numerical simulations. The results provide insight into the connection between basic dynamical flow structures such as vortex tubes and non-Gaussian statistics and characterize the influence of the different field topologies on the single-point statistics.

12:01PM HB.00008 The "Pullin scheme" for non-stationary turbulence , ROBERT RUBINSTEIN, NASA Langley Research Center — The Pullin scheme obtains a solution of the Euler equations from a particle kinetic Boltzmann solution by relaxing the distribution function to a Maxwellian at each time step. We investigate the analog for turbulence closures, using the classical Heisenberg model applied to non-stationary turbulence due to either periodic or linearly increasing forcing as an illustration. Relaxing the spectrum to a local Kolmogorov steady-state spectrum at each time step enforces the Tennekes-Lumley balance between vortex stretching and enstrophy destruction and thereby causes the solution to reproduce the behavior of simple finite dimensional models. We explore the connection between incomplete relaxation and models intermediate in complexity between the full closure and the simplest finite dimensional models.

12:14PM HB.00009 Small-scale intermittency estimations in isotropic homogeneous and anisotropic stably stratified turbulence , SABA ALMALKIE, STEPHEN DE BRUYN KOPS, University of Massachusetts, Amherst — Small-scale intermittency in isotropic homogeneous and anisotropic stably stratified turbulence is examined using high-resolution direct numerical simulations. Statistics of the locally averaged energy dissipation and the scaling exponents of its moments are used as quantitative indicators of intermittency. For the isotropic homogeneous cases, the results are in good agreement with theory and experiments over a range of Reynolds numbers. For the stratified cases, the results help to explain the turbulent bursts observed in stratified flows at low Froude number. Additionally, intermittency estimations based on the single component of the strain rate tensor, a commonly used surrogate, are considered. The differences between the statistical characteristics of the locally averaged surrogate and those of the energy dissipation rate result in overestimating intermittency even in isotropic turbulence. The characteristics of the surrogate in the stratified turbulence are also investigated and the higher level of variability in intermittency estimations in geophysical flows is discussed.

12:27PM HB.00010 Pressure fluctuations and small-scale intermittency in DNS at high resolution[1] , P.K. YEUNG, Georgia Tech, D.A. DONZIS, Texas A&M Univ., K.R. SREENIVASAN, ICTP, Italy — Pressure fluctuations in turbulence are closely related to local flow structure as well as (through the gradients) to the statistics of acceleration which is highly intermittent. We present results from direct numerical simulations of forced isotropic turbulence with 4096^3 grid points, and Taylor-scale Reynolds numbers (R_λ) up to about 1000. For sufficiently high Reynolds number a $k^{-7/3}$ inertial range develops in the pressure spectrum, consistent with experiments. Our present interest is to understand the nature of local flow conditions that correspond to the negative skewmess for the pressure PDF. In particular, conditional statistics show that low pressure is associated with more kinetic energy, larger enstrophy as well as dissipation, i.e. events of strong intermittency which also make accurate sampling of the negative tails of the pressure PDF difficult. By contrast, high pressure is associated with conditions of near-stagnation with strong strain rate but little vorticity. We also observe that the conditionally-averaged dissipation given the pressure also shows much stronger Reynolds number dependence than conditional enstrophy. This is consistent with recent work concerning comparisons between PDFs of dissipation and enstrophy.

[1]Supported by NSF Grant CBET-0553867 and 0553602.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HC Turbulence Simulations V 101C

10:30AM HC.00001 Direct numerical simulation of transonic shock wave/boundary layer interaction , MATTEO BERNARDINI, SERGIO PIROZZOLI, FRANCESCO GRASSO, Dipartimento di Meccanica e Aeronautica, Università di Roma La Sapienza — The interaction of a normal shock wave with a turbulent boundary layer over a flat plate at $M_\infty = 1.3$, $Re_\theta = 1300$ is investigated by means of DNS. The mean flow pattern consists of an upstream fan of compression waves associated with the thickening of the boundary layer and the supersonic region is terminated by a nearly-normal shock foot, which is observed to be significantly bent away from the wall. At the selected conditions the flow does not exhibit separation in the mean. However, the interaction region is characterized by unsteady instantaneous flow reversal in a very large zone, extending for many (upstream) boundary layer thicknesses past the nominal location of the interacting shock, and by the unsteady release of large vortical structures. The adverse pressure gradient tends to suppress the span-wise fluctuations in the near wall region across the interaction, thus yielding two-component isotropic turbulence in the neighborhood of the wall. Frequency-spectra of the wall pressure signature in the interaction region do not reveal the existence of organized low-frequency motions.

10:43AM HC.00002 Analysis of Low-Frequency Unsteadiness in Shockwave and Turbulent Boundary Layer Interaction[1] , STEPHAN PRIEBE, M. PINO MARTIN, Princeton University — The direct numerical simulation (DNS) of a compression-ramp shockwave and turbulent boundary layer interaction is presented. The simulation covers significantly more periods of the characteristic low-frequency shock motion than a previous simulation, which has been validated against experiments.[2,3] We perform a spectral analysis of the flowfield, investigate the behavior of the separation bubble and possible physical mechanisms driving the low-frequency unsteadiness.

[1]Funded by AFOSR Grant # AF/9550-09-1-0464.
[2]Wu, M. & Martin, M. P., AIAAJ, vol.45, pp. 879-889, 2007
[3]Wu, M. & Martin, M. P., JFM, vol.594, pp. 71-83, 2008

10:56AM HC.00003 DNS of transition in supersonic boundary layers[1] , SUMAN MUPPIDI, KRISHNAN MAHESH, University of Minnesota — We are developing the numerical capability to simulate laminar-to-turbulent transition in high speed external flows in complex geometries. Transition to turbulence is associated with increased aerodynamic and thermal loads, and there is a need to reliably predict the behavior and better understand transition mechanisms. We use Direct Numerical Simulation to study transition induced by blowing and suction in a flat plate boundary layer flow at Mach 2.25. We use an unstructured compressible solver with a novel shock capturing scheme that is active only in regions of discontinuities. Preliminary results show good agreement with past work. We will use the simulation results to discuss the time-averaged behavior, transition mechanism, and the importance of simulation details (computational technique, domain size, mesh and timestep).

[1]Supported by AFOSR Grant FA9550-04-1-0341.

11:09AM HC.00004 A Hybrid central difference/WENO scheme to simulate compressible turbulence with shocks and interfaces[1] , ERIC JOHNSEN, JOHAN LARSSON, SANJIVA LELE, Stanford University — Numerical simulations of the late-time turbulent multi-material mixing in the Richtmyer-Meshkov instability (RMI) are challenging due to the contradictory requirements to treat turbulence, for which numerical dissipation must be avoided, and flow discontinuities, for which dissipation is introduced to stabilize the solution. In order to overcome this problem, a hybrid method was developed and is used to selectively apply a non-dissipative scheme in smooth regions, shock capturing at shocks, and interface capturing at interfaces. Results from 1D multicomponent Riemann problems and from 2D single-mode RMI will be presented and compared to other formulations to show that the present multifluid hybrid code does not generate spurious oscillations at discontinuities, that numerical dissipation is contained, and that the total mass, momentum and energy of the system are conserved. Detailed 3D simulations of the interaction of a shock in air with a curtain of SF6 will be presented, and the implementation of diffusive effects will be discussed.

[1]This work is supported by DOE SciDAC (Grant DE-FC02-06-ER25787).

11:22AM HC.00005 Interaction of a spherical shock wave with compressible isotropic turbulence[1] , ANKIT BHAGATWALA, SANJIVA LELE, Stanford University — There have been several studies on the interaction of a planar shock wave with a turbulent inflow. However, such canonical problems do not address the more practical cases of interest, wherein the shock is spherical. We study the interaction of a spherical shock wave expanding outwards into a field of compressible isotropic turbulence. The shock is initiated by depositing a large amount of energy at the center of the domain. As it progresses through the domain, it modifies and is modified by the turbulent flow field. The turbulent field has a high enough turbulent Mach number and Reynolds number to have significant dilatational, vortical and entropic fluctuations. We present two cases, one in which turbulence is significantly modified by the shock and another in which the shock is significantly modified by the turbulence. We identify the physical mechanisms behind these observations and the parameters that are important to this problem. Finally, we highlight some of the crucial differences and challenges of simulating spherical shock-turbulence interaction compared to the planar case.

[1]Supported by DOE-SciDAC

11:35AM HC.00006 DNS and LES of Shock / Isotropic Turbulence Interaction[1] , NATHAN GRUBE, ELLEN TAYLOR, Princeton University, PINO MARTIN, University of Maryland — We use direct numerical simulation (DNS) and large-eddy simulation (LES) to investigate the interaction of highly-compressible isotropic turbulence with nominally planar shock waves. The upstream isotropic turbulence is characterized by turbulence Mach numbers ranging from 0.14 to 0.94 and Taylor microscale Reynolds numbers ranging from 16 to 78. The convection speed of the turbulence through the shock ranges from Mach 1.5 to Mach 5. Streamwise profiles of mean and fluctuating thermodynamic and turbulence quantities are computed along with budgets for Reynolds stresses and fluctuating vorticity. Approximate three-dimensional turbulence energy spectra are computed using Taylor's Hypothesis. Visualization is aided by numerical schlieren animations. The DNS and LES are run using a WENO shock-capturing method in a finite difference code. Subgrid-scale terms are modeled using a dynamic mixed model and an approximate deconvolution model. The LES results are compared with DNS data.

[1]This work is sponsored by AFOSR Grant FA9550-06-1-0323 and NASA CUIP Award NCC3-989.

11:48AM HC.00007 Numerical simulation of aero-optical distortions by turbulent boundary layers[1] , KAN WANG, MENG WANG, University of Notre Dame — Compressible large-eddy simulations are carried out to study the aero-optical distortions caused by flat-plate turbulent boundary layers at $Re_\theta = 1400$ and 2800 and $M = 0.5$. The fluctuating index-of-refraction field is calculated from the density field, and ray tracing is employed to compute the optical path differences (OPD). It is found that optical wavefront distortions are predominantly caused by the logarithmic layer and wake region. Consistent with previous experimental findings, the distortion magnitude is dependent on the direction of propagation due to anisotropy of the boundary-layer vortical structures. An optical beam is distorted more severely when it is tilted toward downstream than upstream. This is explained by a correlation analysis of the fluctuating density field, which shows that the correlation length is larger along downstream-tilted optical paths than upstream-tilted ones. The predicted OPD magnitude and structure at both Reynolds numbers are compared to clarify the Reynolds number dependence and effect of small flow scales.

[1]Supported by HEL-JTO through AFOSR Grant FA 9550-07-1-0504.

12:01PM HC.00008 LES of an inclined jet into a supersonic turbulent crossflow: synthetic inflow conditions , ANTONINO FERRANTE, University of Washington, GEORGIOS MATHEOU, PAUL DIMOTAKIS, California Institute of Technology — The transition and spatial development of a helium sonic jet into a supersonic crossflow (M=3.6) were found to be strongly dependent on crossflow inflow conditions in the Large-Eddy Simulation (LES) of Ferrante *et al.* (AIAA-ASM, 2009-1511). These results indicate that correct turbulent inflow conditions are necessary to predict the main flow characteristics, dispersion and mixing, of a gaseous jet in a supersonic turbulent crossflow. The objective of this work is to provide a methodology for the generation of realistic synthetic inflow conditions for LES of spatially developing, supersonic, turbulent wall-bounded flows. The methodology is applied to the supersonic turbulent flow over a flat wall interacting with an inclined jet matching the experimental conditions of Maddalena *et al.* (JPP, 2006). The sub-grid scale stretched vortex model of turbulent momentum and scalar transport developed by Pullin and co-workers is employed. Inflow turbulence fluctuations are generated by modifying the methodology of Ferrante & Elghobashi (JCP, 2004) to high-Reynolds number, supersonic flows. The results show that the main flow features generated by the gas-dynamic interactions of an inclined jet with the turbulent supersonic crossflow, such as unsteady bow shock, barrel shock, shear layer, and counter-rotating vortex pair, were captured.

12:14PM HC.00009 Direct simulation of heat transfer in a turbulent swept flow over a wire in a channel , REETESH RANJAN, CARLOS PANTANO, University of Illinois at Urbana-Champaign, PAUL FISCHER, ANDREW SIEGEL, Argonne National Laboratory — We present results from direct numerical simulations of heat transfer (considered as a passive scalar) in a turbulent swept flow across a thin, cylindrical wire in a channel. This model mimics the flow through the wire-wrapped fuel pins typical of fast neutron reactor designs. Mean flow develops both along the wire and across the wire, leading to the formation of a turbulent cross-flow regime in the channel. This leads to improvement in heat transfer properties of the channel surface due to enhancement in mixing. The friction Reynolds number in the axial direction is approximately 305. Cross-flow friction Reynolds numbers ranging from 0 to 115 are examined. Two passive scalars at Prandtl number of 1.0 and 0.01 respectively, are simulated in this study. Constant flux boundary conditions are used along the walls of the channel and adiabatic conditions are used along the surface of the wire. The numerical method uses spectral elements in the plane perpendicular to the wire axis and Fourier decomposition in the direction of the axis of the wire. The simulations use up to 107 million collocation points and were performed at the Argonne Leadership BG/P supercomputer. The passive scalar field statistics are investigated, including mean scalar field, turbulence statistics and instantaneous surface scalar distribution.

12:27PM HC.00010 LES of Supersonic Turbulent Channel Flow at Mach Numbers 1.5 and 3[1] , SRIRAM RAGHUNATH, GILES BRERETON, Michigan State University — LES of compressible, turbulent, body-force driven, isothermal-wall channel flows at Re_τ of 190 and 395 at moderate supersonic speeds (Mach 1.5 and 3) are presented. Simulations are fully resolved in the wall-normal direction without the need for wall-layer models. SGS models for incompressible flows, with appropriate extensions for compressibility, are tested $a\ priori$ with DNS results and used in LES. Convergence of the simulations is found to be sensitive to the initial conditions and to the choice of model (wall-normal damping) in the laminar sublayer. The Nicoud–Ducros wall adapting SGS model, coupled with a standard SGS heat flux model, is found to yield results in good agreement with DNS.

[1]We gratefully acknowledge the support of AFRL.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HD Flow Control V 101D

10:30AM HD.00001 The Flow Field on Hydrofoils with Leading Edge Protuberances[1] , DERRICK CUSTODIO, Worcester Polytechnic Institute (WPI), CHARLES HENOCH, Naval Undersea Warfare Center (NUWC), Newport, RI, HAMID JOHARI, California State University, Northridge, OFFICE OF NAVAL RESEARCH COLLABORATION — The exceptional mobility of the humpback whale has been linked to the use of its unique pectoral flippers. Biologists speculate that the flippers leading edge protuberances are a form of passive flow control. Force measurements on 2D hydrofoils with spanwise uniform leading edge protuberances, resembling those seen on the humpback whale flipper, were taken in a water tunnel and have revealed performance modifications when compared to a baseline NACA 63(4)-021 hydrofoil model. Qualitative flow visualization techniques and Particle Image Velocimetry (PIV) flow field measurements on the modified hydrofoils have shown that streamwise vortices originating from the shoulders of the protuberances are the likely cause of performance changes. Varying levels of interaction among adjacent streamwise vortices have been observed as a function of angle of attack and chord location. The circulation of these vortices as a function of angle of attack and spatial location was measured and an analysis of the vortex interactions will be presented.

[1]University Laboratory Initiative (ULI)

10:43AM HD.00002 Aerodynamic Control using Distributed Active Bleed[1] , JOHN M. KEARNEY, GEORGE T.K. WOO, ARI GLEZER, Georgia Institute of Technology — Aero-effected control of lifting surfaces using distributed active bleed driven by surface pressure differences and regulated by integrated louver actuators is investigated in wind tunnel experiments. The interaction between the bleed and cross flows alters the apparent aerodynamic shape of the lifting surface and consequently the distributions of forces and moments. A modular wind tunnel model based on a Clark-Y airfoil is mounted on high-resolution isolated load cells and is equipped with independently-controlled surface louvers on the pressure and suction sides near the leading edge. The louvers are activated using integrated SMA actuators that enable time-dependent actuation. Measurements of time-resolved forces and moment show that the actuation leads to significant changes in the lift force and pitching moment over a range of angles of attack. Phase-locked PIV shows the effects of the bleed on the flow on the suction surfaces and the corresponding changes in the near wake.

[1]Supported by AFOSR.

10:56AM HD.00003 Pseudo Volume-filling Sampling (PVS) via Bouyancy Control in Ocean Systems , ROBERT KROHN, THOMAS BEWLEY, University of California, San Diego — This project address sensing methods for modeling and prediction in structured ocean current flow applications. Well distributed measurements of important flow quantities such as the local fluid velocity, temperature, and pressure are scientifically valuable. Such measurements can be obtained from very simple sensor systems. The challenge is to distribute the sensors uniformly over the volumes of interest. Using a fleet of sensor-equipped free-drifting floats, such as those currently used in the Argo project, uncoordinated distribution has been well documented. The sensor systems are grossly underactuated, and are only capable of controlling their vertical motion, making coordinated distribution difficult. Given that an accurate estimate of the background velocity field is available, however, it is possible to "fly" each individual sensor system, much as a recreational balloonist can direct a balloon fairly accurately by exploiting known velocity shear within the atmosphere. The present work addresses how a network of such underactuated vehicles can be distributed uniformly over the domain of interest using variations in local flow velocities, with both global model predictive control (MPC) algorithms regulated centrally and, in certain well-defined subproblems, simple LQG algorithms implemented locally.

11:09AM HD.00004 A New Sliding Discharge Actuator for Aerodynamic Flow Control , LEE NEUHARTH, FLINT THOMAS — The application of a three-electrode (triode) sliding discharge (TSD) actuator for aerodynamic flow control is examined. The sliding discharge actuator is a hybrid that combines the effects of both a DC corona and dielectric barrier discharge. In the configuration examined in this study, two surface mounted electrodes are placed strategically on the aerodynamic body. One is held at ground and the other is at a large negative DC bias voltage. A third electrode spanning the distance between the two surface electrodes is placed below a dielectric barrier material and is given a high-voltage AC signal. This produces a glow discharge on the aerodynamic surface in the space between the surface electrodes and there is an associated coupling of directed momentum to the ambient fluid that is the basis for flow control. Two example flow control applications of the TSD are considered: (1) the boundary layer on a flat plate and (2) separation control on a symmetric airfoil. Results from both flow control studies are summarized.

11:22AM HD.00005 Aerodynamics of a golf ball with grooves[1] , JOOHA KIM, KWANGMIN SON, HAECHEON CHOI, Seoul National University — It is well known that the drag on a dimpled ball is much lower than that on smooth ball. Choi *et al.* (Phys. Fluids, 2006) showed that turbulence is generated through the instability of shear layer separating from the edge of dimples and delays flow separation. Based on this mechanism, we devise a new golf ball with grooves on the surface but without any dimples. To investigate the aerodynamic performance of this new golf ball, an experiment is conducted in a wind tunnel at the Reynolds numbers of $0.5 \times 10^5 - 2.7 \times 10^5$ and the spin ratios (ratio of surface velocity to the free-stream velocity) of $\alpha = 0 - 0.5$, which are within the ranges of real golf-ball velocity and spin rate. We measure the drag and lift forces on the grooved ball and compare them with those of smooth ball. At zero spin, the drag coefficient on the grooved ball shows a rapid fall-off at a critical Reynolds number and maintains a minimum value which is lower by 50% than that on smooth ball. At non-zero α, the drag coefficient on the grooved ball increases with increasing α, but is still lower by 40% than that on smooth ball. The lift coefficient on the grooved ball increases with increasing α, and is 100% larger than that on smooth ball. The aerodynamic characteristics of grooved ball is in general quite similar to that of dimpled ball. Some more details will be discussed in the presentation.

[1]Supported by the National Research Laboratory Program, MEST.

11:35AM HD.00006 Experimental Investigation of Actuators for Flow Control in Inlet Ducts[1] , JOHN VACCARO, MICHAEL AMITAY, Rensselaer Polytechnic Institute — For military applications, inlet designs are constrained by low observability requirements, which call for the use of an S-duct inlet. The inlets purpose is to limit the line-of-sight to the compressor and decelerate the incoming flow while minimizing total pressure loss, distortion, and unsteadiness. In addition, in unmanned aerial vehicles, the inlet length can determine the overall size of the aircraft. For this reason, aggressive inlets can have a large impact on overall system efficiency. Experiments have been conducted which evaluate the effectiveness of different actuation systems for active flow control in an aggressive S-duct inlet, L/D = 1.5 (at flow conditions representative of flight conditions). Comparisons will be made between: steady and unsteady blowing from a single 2-D tangential slit, spanwise varying injection from a tangential slit, and spanwise varying injection of a hybrid actuator that has both a coanda type injector along with vortex generator jets to eliminate vorticity developed by secondary flow inherent to S-ducts. Evaluations criteria will include total pressure recovery, AIP distortion levels, and unsteady pressure fluctuations.

[1]Project supported by Northrop Grumman Corporation.

11:48AM HD.00007 Numerical Investigation of Actuators for Flow Control in Inlet Ducts , KENNETH JANSEN, MANE, RPI, ONKAR SAHNI, SCOREC, RPI, MICHAEL AMITAY, MANE, RPI — For military applications, inlet designs are constrained by low observability requirements, which call for the use of an S-duct inlet. The inlets purpose is to limit the line-of-sight to the compressor and decelerate the incoming flow while minimizing total pressure loss, distortion, and unsteadiness. In addition, in unmanned aerial vehicles, the inlet length can determine the overall size of the aircraft. For this reason, aggressive inlets can have a large impact on overall system efficiency. Numerical simulations have been carried out which evaluate the effectiveness of steady and unsteady actuation for active flow control in an aggressive S-duct inlet, L/D = 1.5 (at flow conditions representative of flight conditions). These simulations were performed in close co-ordination with the experiments to be able to validate CFD predictions and further provide a complementary and detailed view of the flow field. Comparisons will be made between: steady and unsteady blowing from a single 2-D tangential slit. Evaluations criteria will include total pressure recovery, AIP distortion levels, and unsteady pressure fluctuations. It is noteworthy to mention that the agreement between the CFD predictions and the experimental measurements were found to be very good.

12:01PM HD.00008 3-D Interactions of Synthetic Jets and Cross-Flows - CFD , ONKAR SAHNI, SCOREC, RPI, MICHAEL AMITAY, KENNETH JANSEN, MANE, RPI — The interaction of synthetic jets with a cross-flow was studied numerically at low Reynolds numbers and low angles of attack by using parallel adaptive flow simulations. These investigations were performed in close co-ordination with the experimental studies to precisely match the physical dimensions and parameters. The focus of the work was to explore the details of the flow structures in the vicinity of the synthetic jets. Both instantaneous and phase-averaged flow fields were collected to understand the interactions. It was found that an array of counter-rotating vortical structures formed by the synthetic jets interacts with the cross flow and develops three-dimensionalities as they advect downstream. The effect of the momentum coefficient (or blowing ratio) on the 3-D interaction was also explored. In the case of low momentum coefficient coherent vortical structures were found to be dominant whereas at high momentum coefficient coherent vortical structures breakdown forming random ones. The agreement between the CFD predictions and the experimental measurements were found to be very good.

12:14PM HD.00009 Real-time PIV for Active Flow Control Applications[1] , MATTHEW MUNSON, California Institute of Technology, CHRISTIAN WILLERT, German Aerospace Center (DLR), MORTEZA GHARIB, California Institute of Technology — In pursuit of closed-loop control of vortex formation and separation processes on low-Reynolds number wings, a "real-time" particle image velocimetry system has been developed for use in the low Reynolds number oil tunnel facility at the California Institute of Technology. The use of oil as the working fluid provides slow enough dynamics that the typical latencies inherent to the PIV technique remain manageable. This system allows control decisions to be computed directly from a quantitative evaluation of features in the flow field. The performance of the system will be discussed, along with preliminary efforts toward regulation of vortex shedding processes in the wake of a low-aspect ratio rectangular wing.

[1]Work supported by a Multidisciplinary Research Initiative from the United States Air Force Office of Scientific Research (FA9550-05-1-0369).

12:27PM HD.00010 The Design of Surface Mounted Transducer Arrays for Pressure Field Mapping in Flow Control Applications , MARK POTTER, JONATHAN MORRISON, Imperial College London, GRAHAM ARTHUR, Rutherford Appleton Laboratory — Advances in the field of flow control are leading to increasing demands being placed on pressure based instrumentation systems, in terms of sensitivity and spatial resolution. This work is aimed at accurately mapping pressure fields, in terms of both pressure and pressure-gradient fluctuations at the wall, to provide essential information for closed-loop flow control. To achieve this mapping, a capacitor-based sensor has been developed and constructed using micro fabrication techniques. Using these techniques, an array of robust surface-mounted transducers, with a sensing diameter of 800μm, has been developed with the aim of further size reductions in the future. Presented alongside the structure and fabrication methods for this design of pressure sensor is a description of the electronic systems used to determine the changes in the transducer capacitance, required for accurate sensing, along with preliminary results from a transducer array.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HE Biofluids VI: Cardiovascular 101E

10:30AM HE.00001 Flow Instability and Wall Shear Stress Ocillation in Intracranial Aneurysms , HYOUNGSU BAEK, Brown University, MAHESH JAYAMARAN, Warren Alpert School of Medicine, Brown University, PETER RICHARDSON, GEORGE KARNIADAKIS, Brown University — We investigate the flow dynamics and oscillatory behavior of wall shear stress (WSS) vectors in intracranial aneurysms using high-order spectral/hp simulations. We analyze four patient- specific internal carotid arteries laden with aneurysms of different characteristics : a wide-necked saccular aneurysm, a hemisphere-shaped aneurysm, a narrower-necked saccular aneurysm, and a case with two adjacent saccular aneurysms. Simulations show that the pulsatile flow in aneurysms may be subject to a hydrodynamic instability during the decelerating systolic phase resulting in a high-frequency oscillation in the range of 30-50 Hz. When the aneurysmal flow becomes unstable, both the magnitude and the directions of WSS vectors fluctuate. In particular, the WSS vectors around the flow impingement region exhibit significant spatial and temporal changes in direction as well as in magnitude.

10:43AM HE.00002 Flow patterns and shear stress waveforms in intracranial aneurysms: The effect of pulsatility[1] , FOTIS SOTIROPOULOS, TRUNG LE, IMAN BORAZJANI, University of Minnesota, SAFL TEAM — The wall shear stress on the dome of intracranial aneurysms has been hypothesized to be an important factor in aneurysm pathology and depends strongly on the hemodynamics inside the dome. The importance of patient-specific geometry on the hemodynamics of aneurysms has long been established but the significance of patient-specific inflow waveform is largely unexplored. In this work we seek to systematically investigate and quantify the effects of inflow waveform on aneurysm hemodynamics. We carry out high resolution numerical simulations for an anatomic intracranial aneurysm obtained from 3D rotational angiography (3DRA) data for various inflow waveforms. We show that both the vortex formation process and wall-shear stress dynamics on the aneurysm dome depend strongly on the characteristics of the inflow waveform. We also present preliminary evidence suggesting that a simple non-dimensional number (named the Aneurysm number), incorporating both geometry and inflow waveform effects, could be a good qualitative predictor of the general hemodynamic patterns that will arise in a given aneurysm geometry for a particular waveform.

[1]Minnesota Supercomputing Institute, National Science Foundation and National Institutes of Health

10:56AM HE.00003 Hemodynamic simulations in coronary aneurysms of children with Kawasaki disease[1] , DIBYENDU SENGUPTA, Mechanical and Aerospace Engineering Dept, UCSD, JANE BURNS, Pediatrics Dept, UCSD, ALISON MARSDEN, Mechanical and Aerospace Engineering Dept, UCSD — Kawasaki disease (KD) is a serious pediatric illness affecting the cardiovascular system. One of the most serious complications of KD, occurring in about 25% of untreated cases, is the formation of large aneurysms in the coronary arteries, which put patients at risk for myocardial infarction. In this project we performed patient specific computational simulations of blood flow in aneurysmal left and right coronary arteries of a KD patient to gain an understanding about their hemodynamics. Models were constructed from CT data using custom software. Typical pulsatile flow waveforms were applied at the model inlets, while resistance and RCR lumped models were applied and compared at the outlets. Simulated pressure waveforms compared well with typical physiologic data. High wall shear stress values are found in the narrow region at the base of the aneurysm and low shear values occur in regions of recirculation. A Lagrangian approach has been adopted to perform particle tracking and compute particle residence time in the recirculation. Our long-term goal will be to develop links between hemodynamics and the risk for thrombus formation in order to assist in clinical decision-making.

[1]Burroughs Wellcome Fund

11:09AM HE.00004 Analyzing Transient Turbuelnce in a Stenosed Carotid Artery by Proper Orthogonal Decomposition , LEOPOLD GRINBERG, Brown University, ALEXANDER YAKHOT, Ben-Gurion University of the Negev, GEORGE KARNIADAKIS, Brown University — High resolution 3D simulation (involving 100M degrees of freedom) were employed to study transient turbulent flow in a carotid arterial bifurcation with a stenosed internal carotid artery (ICA). In the performed simulation an intermittent (in space and time) laminar-turbulent-laminar regime was observed. The simulation reveals the mechanism of the onset of turbulent flow in the stenosed ICA where the narrowing in the artery generates a strong jet flow. Time- and space-window Proper Orthogonal Decomposition (POD) was applied to quantify the different flow regimes in the occluded artery. A simplified version of the POD analysis that utilizes 2D slices only - more appropriate in the clinical setting - was also investigated.

11:22AM HE.00005 Numerical Simulation of the Flow in Vascular Grafts for Surgical Applications[1] , PATRICK MCGAH, ALBERTO ALISEDA, University of Washington - Dept. of Mechanical Engineering — Numerical simulation of the human blood vessels, is becoming an important tool in surgical planning and research. Accurate vascular simulations might grant physicians the predictive capability to perform pre-surgical planning. We focus our attention on the implantation of vascular grafts. The high rate of failure of this common vascular interaction is intimately related to the fluid mechanics in the affected region and the subsequent wall tissue remodeling. Here, we will present our current work in developing a methodology for the numerical simulation of vascular grafts which incorporates physiologically realistic geometries and flow boundary conditions. In particular, we seek to correlate the wall shear stress and its spatial (WSSG) and temporal (OSI) variability to wall remodeling as observed in patient specific longitudinal studies. The pulsatility ($Re_{mean} = 800$, $Re_{peak} = 2000$, $Wo = 2$) of the flow gives rise to additional fluid dynamics phenomena such as instability, flow separation, transition, and unsteadiness. Our goal is to describe and evaluate their effect on the wall physiology.

[1]Supported by an R21 grant from NIDDK.

11:35AM HE.00006 The Effect of Aortic Compliance on Left Ventricular Power Requirement , NIEMA PAHLEVAN, MORTEZA GHARIB, California Institute of Technology — Aortic compliance depends on both geometry and mechanical properties of the aorta. Reduction in arterial compliance has been associated with aging, smoking, and multiple cardiovascular diseases. Increased stiffness of the aorta affects the wave dynamics in the aorta by increasing both pulse pressure amplitude and wave speed. We hypothesized that decreased aortic compliance leads to an increased left ventricular power requirement for a fixed cardiac output due to altered pulse pressure and pulse wave velocity. We used a computational approach using the finite element method for solid and fluid domains coupled to each other by using the direct coupling method. A nonlinear material model was used for the solid wall. The fluid flow model was considered to be Newtonian, incompressible, and laminar. The simulation was performed for a heart rate of 75 beats per minute for six different compliances while keeping the cardiac output and the peripheral resistance constant. The results show a trend towards increased left ventricular energy expenditure per cycle with decreased compliance. The relevance of these findings to clinical observations will be discussed.

11:48AM HE.00007 Non-Invasive Measurement of Viscosity in Pulsatile Flow in Elastic Vessels , G.J. BRERETON, M.M. KOOCHESFAHANI, V. DURGESH, Michigan State University — When laminar pulsatile flows in elastic-walled vessels are fully developed, theoretical solutions can be found relating any two flow variables at a given axial location. When the flow variables are both velocities, such as the time-dependent centerline and area-averaged velocities, their interdependence on the wave speed of pulse propagation cancels. When the wave-speed Reynolds number exceeds 1,000, effects of the vessel wall's compliance on the solution become negligible, leaving the vessel radius and the Newtonian viscosity as the only *parameters* in the time-dependent problem. A non-invasive method for determining the fluid viscosity based on such solutions is demonstrated in arbitrarily unsteady flow, that is relevant to the critical problem of accurate *in vivo* measurement of the local viscosity of blood in patients.

12:01PM HE.00008 Mass Transport and Shear Stress as Mediators of Flow Effects on Atherosclerotic Plaque Origin and Growth , RILEY GORDER, ALBERTO ALISEDA, University of Washington — The carotid artery bifurcation (CAB) is one of the leading site for atherosclerosis, a major cause of mortality and morbidity in the developed world. The specific mechanisms by which perturbed flow at the bifurcation and in the carotid bulge promotes plaque formation and growth are not fully understood. Shear stress, mass transport, and flow residence times are considered dominant factors. Shear stress causes restructuring of endothelial cells at the arterial wall which changes the wall's permeability. Long residence times are associated with enhanced mass transport through increased diffusion of lipids and white blood cells into the arterial wall. Although momentum and mass transfer are traditionally coupled by correlations similar to Reynolds Analogy, the complex flow patterns present in this region due to the pulsatile, transitional, detached flow associated with the complex geometry makes the validity of commonly accepted assumptions uncertain. We create solid models of the CAB from MRI or ultrasound medical images, build flow phantoms on clear polyester resin and use an IOR matching, blood mimicking, working fluid. Using PIV and dye injection techniques the shear stress and scalar transport are experimentally investigated. Our goal is to establish a quantitative relationship between momentum and mass transfer under a wide range of physiologically normal and pathological conditions.

12:14PM HE.00009 Evolution of streamwise vortices in a 180° circular bend under physiological flow conditions[1] , FANGJUN SHU, MICHAEL PLESNIAK, George Washington University, CHEKEMA PRINCE, SEAN PETERSON, University of Waterloo — Due to the complex geometry of the vasculature and the pulsatile nature of blood flow, secondary flows are common in blood vessels. Secondary flow vortices within a 180 ° circular bend, under physiological flow conditions, with perturbations introduced by model stents were investigated. Reynolds numbers ranged from 200 to 1400 and the cardiac cycle period was scaled to match the physiological Womersley number, Wo=3.6. LDV and PIV were used to measure the flow fields at different locations in the bend. Companion computations were performed using commercial CFD software. The streamwise velocity profile was skewed toward the inner wall at the entrance of the bend and to shift progressively toward the outer wall further downstream. Counter-rotating Dean vortices were observed during the majority of the cardiac cycle. Even though Wo is relatively low, Lyne vortices were observed to develop after 90 ° of bend, following the systolic peak in the waveform. Because of the strong accelerations, higher frequency harmonic waves are present, which are associated with locally high Wo.

[1]Supported by the National Science Foundation under Grant No. CBET-0909678.

12:27PM HE.00010 Flow in an Aortic Coarctation , LUIS LOMA, PAUL MILLER, JEAN HERTZBERG, University of Colorado, Boulder — Coarctation of the aorta is a congenital cardiovascular defect that causes a constriction in the descending thoracic aorta. To gain a better understanding of the cause of post-surgical problems, a rigid glass and a compliant *in vitro* model of the aortic arch and descending aorta with a coarctation were constructed. Near-physiologic compliance was obtained using a silicone elastomer. Stereoscopic PIV was used to obtain 3D velocity maps. Results show a high speed turbulent jet formed at the exit of the coarctation. Flow in the rigid model was significantly different from in the compliant model. In the rigid model, the jet was symmetric, creating a toroidal recirculation area. In the compliant model, the jet was directed towards the medial wall, inducing flow reversal only at the lateral wall. Peak velocities and turbulence intensities were higher in the rigid model, however shear rate values in the compliant model were significantly above both the rigid model and normal *in vivo* values at the medial wall. In both models the reattachment region fluctuated, creating oscillatory shear.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HF Microfluidics: Particles 101F

10:30AM HF.00001 Directed assembly in designed flow fields , TOBIAS M. SCHNEIDER, SHREYAS MANDRE, MICHAEL P. BRENNER, School of Engineering and Applied Sciences, Harvard University — Various approaches aim at the controlled assembly of mesoscopic objects intro predefined shapes. While self-assembling strategies are conceptually difficult and often hard to implement, we present a new concept for directed assembly. We show that by controlling the fluxes on the boundary of a Hele-Shaw cell the flow field in the cell can be tailored such that many particles are simultaneously advected along prescribed trajectories and thereby assembled in the desired shape. Optimizing the particle paths allows to reduce the required fluxes such that they should be experimentally accessible using microfluidic techniques.

10:43AM HF.00002 Concentration distributions of arbitrary shaped particles in microfluidic channel flows , ARVIND SAIBABA, ERIC SHAQFEH, ERIC DARVE, Stanford University — We are interested in the study of the transient and steady state concentration distribution of orientable Brownian particles across channels at low Reynolds numbers. This is important in understanding margination of blood "particles" including platelets as well as new drug delivery and cancer nanotechnology particles which are involved in hemostasis as well as delivering drugs to the vascular endothelial cells. Although our formulation is general, the particles we consider are rigid Brownian "surfboards" which have been found to be effective in drug delivery since they are resistant to leukocyte attack [1]. The Stokes flow in the channel around the particles, driven by a mean pressure gradient, is computed using the Boundary Element method within the single layer formulation. The particle motion is calculated using rigid body dynamics with a contribution due to Brownian motion that satisfies the Fluctuation-Dissipation theorem. Finite concentrations are considered, and all hydrodynamic interactions are included. The concentration distribution is computed and interpreted as a balance between the concentration dependent variation in the non-equilibrium particle osmotic pressure and the cross stream particle normal stresses.

[1] J. A. Champion, S. Mitragotri, "Role of target geometry in phagocytosis", PNAS 103, 4930-4934, (2006)

10:56AM HF.00003 Transmitting chemical and mechanical signals via a cluster of microcapsules , AMITABH BHATTACHARYA, GERMAN V. KOLMAKOV, ANNA C. BALAZS, Dept of Chem Engg, University of Pittsburgh — Biological cells often perform tasks collectively by sensing the local density of cells and then performing a particular task in concert (e.g. emitting light) when this cell density increases above a certain threshold. Using an approach based on the Lattice-Boltzmann method, we simulate a similar synthetic system consisting of primarily two kinds of signaling microcapsules, immersed in a fluid, and sitting on an adhesive surface. The first kind constantly releases "agonist" molecules, and the second kind release nanoparticles above a certain threshold concentration of "agonist" molecules. The nanoparticles adsorb onto the surface and decrease the capsule-surface adhesion strength at the point of adsorption. The resulting gradients in adhesion strength along the surface induces motion in the microcapsules. We examine arrangements of these microcapsules in which mechanical and chemical signals can cascade through a cluster of microcapsules, and comment on the robustness of this system.

11:09AM HF.00004 Collective behavior and quorum sensing in a system of communicating microcapsules , GERMAN KOLMAKOV, AMITABH BHATTACHARYA, ANNA BALAZS, University of Pittsburgh, PROF. ANNA C. BALAZS GROUP TEAM — We report the results on collective motion of polymeric microcapsules in a fluid-filled microchannel. We consider the case where motion of the nanoparticle-filled microcapsules is controlled by adhesion at the channel's wall and hydrodynamic coupling between the capsules. Using the hybrid Lattice Boltzmann method for fluid dynamics and Lattice spring model for the micromechanics of elastic solid, we determined how the characteristics of the substrate, the polymeric shell, encapsulated fluid and the surrounding solution affect the capsule's velocity and "gait" of the capsule within the system. In numerical computations we find the conditions under which microcapsules communicating through modification of the microchannel surface by released nanoparticles exhibit collective motion, thereby mimicking behavior of the colony of living cells. In particular, we show that this system demonstrates a quorum sensing. That is, the capsules motion depends on population and behavior of neighboring groups of capsules. Finally, the design of a repair-and-go system is presented, in which we show that deposition of nanoparticles from moving microcapsules onto a damaged substrate can be used as an effective tool for selective repair of defects or cracks on the substrate.

11:22AM HF.00005 Microfluidic Manipulation of Suspended Single Cells: Cell Deformation and Mechanical Stress Analysis , NATHALIE NEVE, SEAN KOHLES, DEREK TRETHEWAY, Portland State University — Bone and cartilage cells experience multiple stresses in vivo. The optimum mechanical conditions for cell health are not fully understood. With the recent development of an integrated optical tweezer with micron resolution particle image velocimetry, the opportunity to apply and measure controlled multiaxial stresses to suspended single cells is possible. In this work, we examine optically suspended cells in uniform and extensional flow fields. The cellular deformation and applied fluid induced stresses are determined. Maximum applied stresses for uniform flows are substantially smaller than typical fluid stresses in cell monolayer studies. Extensional flows enable potentially higher applied stresses. For extensional flows, bone cells show no deformation for shear stresses up to 250 mPa, while significant deformation of muscle cells is observed.

11:35AM HF.00006 Directed locomotion of bimetallic synthetic nanomotors , JONATHAN POSNER, PHILIP WHEAT, JEFFREY MORAN, CHRISTINE BURDETT, J. BURDICK, Arizona State University, R. LAOCHAROENSUK, Laocharoensuk, P. CALVO-MARZAL, K. MANESH, D. KAGAN, S. BALASUBRAMANIAN, M. CARDONA, G.-U. FLECHSIG, JOE WANG, UC San Diego — Controlled motion of synthetic nanoscale motors may represent a major step towards the development of practical nanomachines and autonomous microsystems. Bimetallic nanorods can autonomously propel themselves at hundred of body lengths per second through aqueous solutions by using hydrogen peroxide as a fuel. The magnetic and chemical controlled motion of Pt-Ni-Au nanorods is presented. The magnetic properties of nickel-loaded nanomotors offer controlled cargo manipulations, including en-route load, drag and release of spherical cargo that have volumes two orders of magnitude larger than the nanomotors itself. Nanomotors can be directed through microfluidic channel networks and motion triggered using locally generated chemical species. The nanomotor locomotion force is determined by measuring the velocity of motors towing spherical cargo. The nanomotors approximately generate 0.16 picoNewtons of force and can transport microscale cargo with a coefficient of Stokes drag that is eight times their own.

11:48AM HF.00007 Locomotion of a rod-shaped nanomotor propelled by heterogeneous catalytic reactions , JEFFREY MORAN, Mechanical Engineering, Arizona State University, JONATHAN POSNER, Mechanical Engineering, Chemical Engineering, Arizona State University — Bimetallic nanorods can autonomously propel themselves through aqueous solutions by using hydrogen peroxide as a fuel. Several physical arguments have been proposed to describe the physics underlying the chemically-powered locomotion of these synthetic nanomotors, but there is no accepted or detailed theory on the propulsion mechanism. A computational simulation and scaling analysis of rod-shaped nanoparticles with asymmetric surface fluxes is presented. The model shows that locomotion is driven by electric body forces that arise due to finite space charge and internally generated electric fields surrounding the rod. The electric fields and charge density are generated by dipolar cation fluxes, such as those generated by heterogeneous electrochemical reactions. The scaling analysis and detailed simulations predict that the nanomotor velocity depends on the reaction flux, nanorod electrical surface potential, solvent viscosity, and rod geometry. Strong agreement is observed between the scaling analysis and simulations.

12:01PM HF.00008 Confinement-induced partial screening of intra-chain hydrodynamic interactions[1] , YENG-LONG CHEN, JEN-FANG CHANG, PO-KENG LIN, Institute of Physics, Academia Sinica — In studies of polymer dynamics in a confined environment, one of the most commonly invoked assumptions is that intra-chain hydrodynamic interactions are completely screened when the confinement length is smaller than the polymer radius of gyration (Rg). However, recent experiments of single DNA molecules confined in nanoslits have observed that chain diffusivity (D) dependence on slit height (H) does not follow the Rouse scaling exponent of 2/3. Rather, the scaling exponent is found to be around 0.5 for Lk $<$ H $<$ Rg, where Lk is the DNA Kuhn length. This suggests that intra-chain hydrodynamic interactions are not completely screened. In this work, we carried out experiments and complementary simulations to characterize the effect of screening on polymer dynamics. To model the polymer dynamics, we employ a method of Brownian dynamics, combined with the lattice Boltzmann method, to simulate polymer dynamics confined in nanochannels. We find that hydrodynamic screening occurs gradually as the channel height decreases, and also that complete screening in H $<$ Lk channels leads to topological independence of chain diffusivity.

[1]We acknowledge support from NSC 96-2112-M-001-039-MY3 and 95-2112-M-001-051-MY3.

12:14PM HF.00009 Three-dimensional x-ray imaging of macro-clusters in ferrofluids , WAH-KEAT LEE, Argonne National Laboratory — Ferrofluids are a class of magnetic fluids where nano-sized ($\sim$ 10 nm) magnetic particles are dispersed in a carrier fluid. Ferrofluids have long been used for vacuum seals, but lately, has been proposed for a multitude of new applications including heat transfer and biomedicine. It has been known for some time that the magnetic particles tend to align with an applied magnetic field and that the individual chains can coalesce and form thick and long macro-sized structures whose shapes depend on the properties of the ferrofluid and the applied field. However, due to their opacity to visible light, ferrofluid experiments have been mainly limited to very thin films ($\sim$ 10s of microns). Since the macro-structures can be in the 10-100 micron range, thin film measurements are susceptible to wall effects. TEM and resin techniques have been used to study the structure of these clusters. However, it is doubtful if these frozen or dried structures reflect the natural fluid state. Here, we present x-ray microtomography measurements on a mm-sized tube of ferrofluid under an applied magnetic field. We show the three-dimensional nature of the columns and labyrinth structures. The measurements also allow us to provide estimates on the local magnetic particle concentration within the ferrofluid.

12:27PM HF.00010 Direct measurement of shear-induced cross-correlation of Brownian motion , ANDREAS ZIEHL, CHRISTIAN WAGNER, Universität des Saarlandes — Shear-induced cross-correlations between particle fluctuations perpendicular and along streamlines are investigated experimentally and theoretically in a linear shear flow. We used optical tweezers to localize one or two particles, each in a harmonic potential, and to detect the positions of the particles as a function of time with a high spatial precision below 8nm. These positions are recorded via a high speed camera with 15kHz resolution. In contrast to measurements in a quiescent fluid, we find that in shear flow, generated in a special designed micro fluidic device, orthogonal movements of a bead in stream- and gradient- directions are correlated and the time reversal symmetry is broken. Again in a quiescent fluid, fluctuations of two particles, separated by a few microns, are known to be anti-correlated along their connecting vector due to hydrodynamic coupling. In linear shear flow, we found a coupling process that correlates the orthogonal directions of the two particles. The correlation exhibits a minimum in time and again the time reversal symmetry is broken.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HG Rarefied Gases and Direct Simulation Monte Carlo (DSMC) 101G

10:30AM HG.00001 DSMC Predictions of Chemical Reaction Rates between Atmospheric Species , M.A. GALLIS, R.B. BOND, J.R. TORCZYNSKI, Sandia National Laboratories — A recently proposed chemical reaction model based solely on molecular-level information is applied to calculate equilibrium and non-equilibrium chemical reaction rates for atmospheric reactions in hypersonic flows. The DSMC model is capable of reproducing measured equilibrium reaction rates without using any macroscopic reaction-rate information. Since it uses only molecular-level properties, the new model is inherently able to predict reaction rates for arbitrary non-equilibrium conditions. The DSMC-predicted chemical reaction rates are compared to theoretically calculated and experimentally measured reaction rates for non-equilibrium conditions. The observed agreement provides strong evidence that molecular-level modeling of chemical reactions provides an accurate method for predicting equilibrium and non-equilibrium chemical reaction rates. 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.

10:43AM HG.00002 Direct Simulation Monte Carlo Investigation of Noncontinuum Couette Flow , J.R. TORCZYNSKI, M.A. GALLIS, Sandia National Laboratories — The Direct Simulation Monte Carlo (DSMC) method of molecular gas dynamics is used to study noncontinuum effects in Couette flow. The walls have equal temperatures and equal accommodation coefficients but unequal tangential velocities. Simulations are performed for near-free-molecular to near-continuum gas pressures with accommodation coefficients of 0.25, 0.5, and 1. Ten gases are examined: argon, helium, nitrogen, sea-level air, and six Inverse-Power-Law (IPL) gases with viscosity temperature exponents of 0.5, 0.6, 0.7, 0.8, 0.9, and 1.0, as represented by the Variable Soft Sphere (VSS) interaction. In all cases, the wall shear stress is proportional to the slip velocity. The momentum transfer coefficient relating these two quantities can be accurately correlated in terms of the Knudsen number based on the wall separation. The two dimensionless parameters in the correlation are similar for all gases examined. 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.

10:56AM HG.00003 A multiscale, multiphysics simulation method for rarefied gas flows , DAVID KESSLER, ELAINE ORAN, CAROLYN KAPLAN, Naval Research Laboratory — We present a coupled multiscale, multiphysics solution algorithm, CM^3, for rarefied gas flows. The method uses a general solution of the compressible fluid-dynamics equations that incorporates stresses and heat fluxes calculated directly using the Direct Simulation Monte Carlo (DSMC) method. The CM^3 is designed to solve transition-regime flows at a much lower computational cost than possible by directly solving the Boltzmann equation. The CM^3 is tested on a low-speed, Rayleigh flow and a thermal Fourier flow for several Knudsen numbers. Velocity, temperature, shear stress, and heat flux profiles compare well with DSMC solutions. We discuss the algorithmic details that are necessary to implement a true multiscale method, building upon the conceptual framework of E & Engquist's (2003) heterogeneous multiscale methods.

11:09AM HG.00004 Data Structures and Adaptive Mesh Refinement for a 3-Level Embedded Cartesian Mesh DSMC Implementation , CHONGLIN ZHANG, DA GAO, TOM SCHWARTZENTRUBER, University of Minnesota — The data structures and overall algorithms of a newly developed 3-D direct simulation Monte Carlo (DSMC) program are outlined. The code employs an embedded 3-level Cartesian mesh, accompanied by a cut-cell algorithm to incorporate triangulated surface geometry into the adaptively refined Cartesian mesh. Such an approach enables decoupling of the surface mesh from the flow field mesh, which is desirable for near-continuum flows, flows with large density variation, and also for adaptive mesh refinement (AMR). Two separate data structures are proposed in order to separate geometry data from cell and particle information, leading to high scalability and efficient AMR for parallel simulations. A simple and efficient AMR algorithm that maintains local cell size consistent with the local mean-free-path and therefore a constant number of particles in each cell will be detailed. The 3-level embedded Cartesian mesh combined with AMR allows increased flexibility for precise control of local mesh size and time-step, both vital for accurate and efficient DSMC simulation. Verification and validation of the code will be provided, and DSMC results for 3-D flows with large density variations will also be presented.

11:22AM HG.00005 Parallel Performance Optimization of the Direct Simulation Monte Carlo Method , DA GAO, CHONGLIN ZHANG, THOMAS SCHWARTZENTRUBER, University of Minnesota at Twin Cities — Although the direct simulation Monte Carlo (DSMC) particle method is more computationally intensive compared to continuum methods, it is accurate for conditions ranging from continuum to free-molecular, accurate in highly non-equilibrium flow regions, and holds potential for incorporating advanced molecular-based models for gas-phase and gas-surface interactions. As available computer resources continue their rapid growth, the DSMC method is continually being applied to increasingly complex flow problems. Although processor clock speed continues to increase, a trend of increasing multi-core-per-node parallel architectures is emerging. To effectively utilize such current and future parallel computing systems, a combined shared/distributed memory parallel implementation (using both Open Multi-Processing (OpenMP) and Message Passing Interface (MPI)) of the DSMC method is under development. The parallel implementation of a new state-of-the-art 3D DSMC code employing an embedded 3-level Cartesian mesh will be outlined. The presentation will focus on performance optimization strategies for DSMC, which includes, but is not limited to, modified algorithm designs, practical code-tuning techniques, and parallel performance optimization. Specifically, key issues important to the DSMC shared memory (OpenMP) parallel performance are identified as (1) granularity (2) load balancing (3) locality and (4) synchronization. Challenges and solutions associated with these issues as they pertain to the DSMC method will be discussed.

11:35AM HG.00006 Molecular Dynamics Simulations of Nanoscale Gas Flows , MURAT BARISIK, BOHUNG KIM, ALI BESKOK, Old Dominion University — Three-dimensional molecular dynamics (MD) simulations of rarefied gas flows confined within nano-scale channels are investigated by introduction of a smart wall model that drastically reduces the memory requirements of MD simulations for gas flows. The smart wall molecular dynamics (SWMD) represents three-dimensional FCC walls using only 74 wall molecules. Linear Couette flow of argon at Knudsen number 10 is investigated using the SWMD utilizing Lennard-Jones potential interactions. Presence of the walls creates an additional length scale based on the Lennard-Jones force field near the walls. This is typically 3 molecular diameters (σ) in the parametric regime studied here. Therefore 3σ region near the walls becomes a critical length scale that admits deviations from the kinetic theory. Our results have shown increase in the gas density and sudden change of the velocity profiles within this region. Kinetic theory solutions based on the Boltzmann equation neglect the wall force field effects, and hence, cannot predict this density increase and the change in the velocity profile. We have shown that the velocity profile within the interface region exhibits self similar behavior regardless of the channel height (provided that $H > 6\sigma$). Overall, the slip velocity is over predicted using kinetic theory solutions.

11:48AM HG.00007 Catalytic Mechanism Modeling of Oxygen/Platinum Systems using ReaxFF MD Simulation , PAOLO VALENTINI, THOMAS SCHWARTZENTRUBER, University of Minnesota, IOANA COZMUTA, Eloret Corporation — ReaxFF Molecular Dynamics simulations are performed to study some of the fundamental surface mechanisms that characterize the catalytic behavior of a Pt(111) surface exposed to oxygen. The use of the reactive empirical potential ReaxFF allows the simulation of chemical bond breaking/formation, essential to describe the detailed processes at the surface, while maintaining computational feasibility for rather massive systems (thousands of atoms). The ReaxFF potential was initially trained using a set of Quantum Chemistry energy data relevant for the system of interest, including molecular/atomic adsorption enthalpies at various surface sites. Subsequently, the sticking coefficients were determined for oxygen molecules impinging onto the surface. The MD simulations well reproduce some of the experimental trends observed for the adsorption process in molecular-oxygen/platinum systems. At very low incident energies (less than 0.1 eV), the adsorption is determined by a weakly-bound physisorption state, but is rapidly suppressed by additional rotational energy (due to steric hindrance) or by increasing the substrate temperature. At higher incident energies, the sticking probability levels off due to dynamic trapping as observed experimentally. Because ReaxFF is solely based on Quantum Chemistry, the objective of this study is to extend the approach proposed here to other gas/surface systems for which good experimental evidence on the detailed catalytic surface processes is currently not available.

12:01PM HG.00008 A robust semi-discrete quadrature-based moment method for solution of Boltzmann equation , PRAKASH VEDULA, University of Oklahoma — We present an efficient and robust semi-discrete quadrature-based moment method (sDQMOM) for solution of the Boltzmann equation with the full collision operator, for prediction of nonequilibrium flows in the rarefied thru continuum regimes. In the sDQMOM formulation, the distribution function is represented by a set of delta functions with associated weights and locations, whose evolution is determined by semi-discrete equations obtained from evolution of macroscopic moment equations in conservative form, where spatial fluxes of moments are evaluated using appropriate flux limiters. Based on this representation of the distribution function, the generalized moment production terms due to full collision operators can be evaluated analytically using multinomial expansions that ensure preservation collision invariants and correct rates of evolution of selected low order moments. It appears that sDQMOM also addresses some previously encountered limitations of quadrature-based moment methods arising from discontinuities and the ill-posed problem of moment inversion. To ensure realizability and robustness, an automatic and adaptive moment constraint selection algorithm for sDQMOM is also proposed based on orthogonal polynomials with a discrete support. Prediction capabilities of sDQMOM are demonstrated via good agreement between sDQMOM and other approaches for Poiseulle flow and shock tube problem.

12:14PM HG.00009 ABSTRACT WITHDRAWN –

12:27PM HG.00010 ABSTRACT HAS BEEN MOVED TO PU.00005 –

Monday, November 23, 2009 10:30AM - 12:40PM –

Session HH Drops V: Drops and Surfaces 101H

10:30AM HH.00001 On the collision and mixing of water droplets on superhydrophobic surfaces , MICHAEL A. NILSSON, JONATHAN P. ROTHSTEIN, University of Massachusetts Amherst — The dynamics of water drop collisions on superhydrophobic surfaces is investigated using high-speed photography. Teflon is sanded to create the superhydrophobic surfaces. The results of the surface fabrication technique are presented, showing the effect of grit size on hysteresis. This method of creating superhydrophobic surfaces allows for the specification of varied advancing contact angles with similar hysteresis, or varying hysteresis with near similar advancing contact angles. Deionized water droplets are made to collide on these surfaces by propelling one droplet into another using a burst of pressurized air. The subsequent collision is captured, and several impact characteristics are calculated as a function of contact angle hysteresis. The Weber number and impact number are calculated, as well as the maximum deformation of the combined drop. In some experiments, the drops left the surface after collision even with low hysteresis at the low Weber numbers tested. Characteristic images of different regimes of the collision dynamics will be presented, as will how each of these regimes affect the mixing of the drops.

10:43AM HH.00002 Spontaneous Jumping of Coalescing Drops on a Superhydrophobic Surface , JONATHAN BOREYKO, CHUAN-HUA CHEN, Duke University — When micrometric drops coalesce in-plane on a superhydrophobic surface, a surprising out-of-plane jumping motion was observed. Such jumping motion triggered by drop coalescence was reproduced on a Leidenfrost surface. High-speed imaging revealed that this jumping motion results from the elastic interaction of the bridged drops with the superhydrophobic/Leidenfrost surface. Experiments on both the superhydrophobic and Leidenfrost surfaces compare favorably to a simple scaling model relating the kinetic energy of the merged drop to the surface energy released upon coalescence. The spontaneous jumping motion on water repellent surfaces enables the autonomous removal of water condensate independently of gravity; this process is highly desirable for sustained dropwise condensation.

10:56AM HH.00003 Scaling of anisotropic droplet shapes on chemically stripe-patterned surfaces , STEFAN KOOIJ, OLESYA BLIZNYUK, ELIZAVETA VERESHCHAGINA, BENE POELSEMA, IMPACT and MESA+ Institutes, University of Twente — We present experimental results on the tunable anisotropic wetting behavior on chemically patterned anisotropic surfaces. The equilibrium shape of asymmetric glycerol droplets, arising from patterns of alternating hydrophilic (pristine SiO_2) and hydrophobic (fluoroalkylsilane self-assembled monolayers) stripes with dimensions in the low-micrometer range, are investigated in relation to the stripe widths. Owing to the well-defined small droplet volume, the equilibrium shape as well as the observed contact angles exhibit unique scaling behavior. Only the relative width of hydrophilic and hydrophobic stripes proves to be a relevant parameter. Our results on morphologically flat, chemically patterned surfaces show similarities with those of experiments on topographically corrugated substrates. They are discussed in terms of the energetics at the liquid-solid interface.

[1] O. Bliznyuk, E. Vereshchagina, E.S. Kooij, B. Poelsema, Phys. Rev. E **79** (2009) 041601

11:09AM HH.00004 Initial spreading kinetics of high-viscosity droplets on anisotropic surfaces , OLESYA BLIZNYUK, STEFAN KOOIJ, BENE POELSEMA, University of Twente — Liquid droplets on chemically patterned surfaces consisting of alternating hydrophilic and hydrophobic stripes exhibit an elongated shape [1]. To assess the kinetics we present experimental results on the spreading of glycerol droplets on such surfaces using a high-speed camera. Two spreading regimes are observed, expressed in terms of the time-dependent droplet base diameter which can be described by a $r(t) \propto t^n$ power law. Initially, in what is referred to as the inertial regime, the kinetics is dominated by the liquid, and spreading is only weakly dependent on the specific surface properties. As such, liquid spreading is isotropic and the contact line maintains a circular shape. Our results reveal a remarkably long inertial regime, as compared to previous results and available models. Subsequently, in the viscous regime, interactions between the liquid and underlying pattern govern the dynamics. The droplet distorts from a spherical cap shape to adopt an elongated morphology that corresponds to the minimum energy configuration on stripe-patterned surfaces.

[1] O. Bliznyuk, E. Vereshchagina, E.S. Kooij, B. Poelsema, Phys. Rev. E 79 (2009) 041601

11:22AM HH.00005 Numerical studies of relationship between splash and dynamic contact angle , KENSUKE YOKOI, VLADIMIR MITKIN, THEO THEOFANOUS, UCSB — We numerically studied splashes on dry surfaces in terms of dynamic contact angle. Our simulation model based on CLSVOF (Coupled level set and Volume-of-Fluid) method, CIP (constraint interpolation profile) method and VISAM3 (Volume/Surface Integrated Average based Multi-Moment Method) has capability to simulate splashes in droplet impact onto flat solid surface. In our simulations, we varied only dynamic advancing contact angle as all other parameters are fixed including equilibrium contact angle. The numerical results show that, as the dynamic advancing contact angle increases, the outer rim of the liquid lamella becomes thick and it seems the number of fingers reduces. On the other hand, we only varied equilibrium angle. However, the equilibrium angel hardly affects the behavior of splashes. We believe that dynamic advancing contact angle is one of key parameters of splashing. We also compared the numerical results with several existing experimental data.

11:35AM HH.00006 Influence of Liquid Type on Drop Impingement on Rib and Cavity Superhydrophobic Surfaces , JOHN PEARSON, DANIEL MAYNES, BRENT WEBB, Brigham Young University — We report results of an experimental investigation of liquid drops impinging on superhydrophobic surfaces. The surfaces are fabricated in Silicon wafers with micro-ribs and cavities (grooves) that are coated with a fluoropolymer or teflon hydrophobic coating. Liquid droplets of known size were dropped from heights ranging from 0.5 to 50 cm onto the surfaces and the pre-impingement freefall, surface impact, and droplet deformation were imaged at a rate of 6000 frames/second with a digital camera. The droplets were either water, ethanol, or a glycerine/water mixture. The droplet impact speed, maximum droplet spread, horizontal spread speed, vertical speed of the issuing jet, and the time between impact and formation of the issuing jet were all characterized. The results show that the overall impact dynamics are strongly influenced by the different impinging surface conditions and the fluid type. Results were compared with previously proposed analytical models and suggestions for improving those models are made.

11:48AM HH.00007 Insight into drop runback on hydrophilic to superhydrophobic surfaces by shearing airflow , ANDREW J.B. MILNE, ALIDAD AMIRFAZLI, Department of Mechanical Engineering, University of Alberta, Edmonton, AB, T6G 2G8, Canada — Drop runback has many diverse applications including airfoil icing and fuel cell flooding. In this talk, we use surface science and fluid dynamics principles to explain incipient runback for a drop exposed to shearing airflow. Through experiments with single drops of water and hexadecane (0.5-100 μl) on PMMA, Teflon, and a superhydrophobic aluminum surface (SHS), wetting parameters such as surface tension, drop shape and contact angle are found to be major controllers of the minimum required air velocity for drop shedding. Exponential functions are proposed that relate air velocity to drop base length and projected area. By normalizing the results, the three water systems can be collapsed to a single curve that also explains results from other researchers, vastly increasing predictive power. SHS are seen to shed drops more easily compared to the other surfaces, with evidence that the drops roll along the surface instead of sliding. Using high speed video, oscillating drop shape and variation of contact angles are also analyzed as they change with air and drop speed.

12:01PM HH.00008 The Effect of Hydrogen Passivation Surface on Silicon Nanodroplets Coalescence , TAKUMI HAWA, The University of Oklahoma — Understanding a fundamental formation mechanism of nanoparticles growth and controlling primary particle size and extent of agglomeration when grown from the gas–phase are the significant challenges in the use of nanoparticles. In this talk a possibly mathematical model to describe the droplet coalescence is presented. Here the coalescence of hydrogen terminated silicon surface slowing the process has been studied, and results are compared with molecular dynamics simulations. Nanodroplets of the size between 2 and 6 *nm* at 1500 K were considered. The hydrogen passivation surface completely changes and slows the beginning of the coalescence process. In addition, the presence of hydrogen atoms reduces surface tension of the droplet about 40 to 50%. The model is able to describe both initial induction period and the standard coalescence period. It presents that the effective surface tension decreases with increasing hydrogen coverage, making it harder for droplets to coalesce. http://www.ou.edu/mms/

12:14PM HH.00009 Two-dimensional droplet spreading over random topographical substrates , NIKOS SAVVA, Department of Chemical Engineering, Imperial College London, UK, GREG PAVLIOTIS, Department of Mathematics, Imperial College London, UK, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, UK — We examine theoretically the effects of a random topographical substrate on the motion of a two-dimensional droplet by developing appropriate statistical approaches. Our theory is based on a set of integro-differential equations for the two droplet fronts, previously obtained for deterministic substrates through a singular perturbation method. We provide a stochastic representation of random substrates as families of certain stationary random functions parametrized by a characteristic amplitude and a characteristic wavenumber. The droplet footprint is found to be a normally distributed random variable as it evolves towards equilibrium. The statistical analysis of the droplet shift along the substrate is highly non-trivial, but its variance can be deduced theoretically at early times and in the long-time limit. It is shown that substrate roughness tends to decrease the wetting properties of the droplet and that its approach to equilibrium is significantly slower for the droplet shift than its footprint, suggesting that the droplet has the tendency to slide without spreading along the substrate features in search for equilibrium. Our theoretical predictions are verified by numerical experiments.

12:27PM HH.00010 Effect of contact angle and humidity on evaporation of inkjet-printed colloidal drops , YING SUN, Drexel University, VADIM BROMBERG, SAILEE GAWANDE, TIMOTHY SINGLER, Binghamton University — Inkjet printing has attracted much attention in recent years due to its ability to dispense precise amounts of functional materials onto targeted areas. Although evidence exists for a multi-stage evaporation of a sessile drop, the actual evaporation behavior of an inkjetted colloidal drop is not well understood. In this study, a novel visualization technique is developed wherein aqueous suspensions of fluorescent particles are inkjetted onto transparent surfaces and the evaporation dynamics are observed in real-time using a high-power microscope. Two influencing parameters, the ambient humidity and substrate wettability, are systematically varied. It has been confirmed that jetted drops follow a *pinned*, *dewetting*, and *mixed* multi-stage evaporation process. The results also show that the relative humidity acts mainly to accelerate or decelerate the process whereas its relationship to contact angle is not as direct. Contact angle hysteresis plays an important role in controlling the initial pinned mode. For lower contact angle substrates, evaporation drives a flow of particles to deposit near the contact line which set the conditions for the dewetting stage that follows. Finally, a diffusion-controlled evaporation model is used to predict the time internals for each evaporation stage. The model agrees well with the experimental data, especially for the dewetting mode.

Monday, November 23, 2009 10:30AM - 12:40PM –

Session HJ Bubbles V: General Experiments 101I

10:30AM HJ.00001 Superheated water drops in hot oil , ENRIQUE SOTO, Penn State University, ROBERTO ZENIT, Universidad Nacional Autónoma de México, ANDREW BELMONTE, Penn State University — Drops of water at room temperature were released in hot oil, which had a temperature higher than that of the boiling point of water. Initially, the drop temperature increases slowly mainly due to heat transfer diffusion; convective heat transfer is small because the motion takes place at a small Reynolds number. Once the drop reaches the bottom of the container, it sticks to the surface with a certain contact angle. Then, a part of the drop vaporizes: the nucleation point may appear at the wall, the interface or the bulk of the drop. The vapor expands inside the drop and deforms its interface. The way in which the vapor expands, either smooth or violent, depends on the location of the nucleation point and oil temperature. Furthermore, for temperatures close to the boiling point of water, the drops are stable (overheated); the vaporization does not occur spontaneously but it may be triggered with an external perturbation. In this case the growth of the vapor bubble is rather violent. Many visualization for different conditions will be shown and predictions of the growth rate will be discussed.

10:43AM HJ.00002 Numerical Analysis of Optical Fiber Probing by Ray Tracing Method. , AKIHIRO SAKAMOTO, Sumitomo Metal Indudries, Co., Ltd., TAKAYUKI SAITO, Shizuoka Univ. — Ray tracing numerical analysis of optical fiber probing, which is one of practical measurement techniques of bubbles and droplets dynamics, has been developed as a reliable instrument of bursting gas-liquid surface. The analysis is constructed by bounded surfaces of flat planes, cylinders, and ellipsoids and constant refractive index objects in three dimensions. The rays' energy of reflection and refraction are calculated repeatedly on the surfaces considered with polarization. The rays' source is assumed as linear polarized light with random fluctuations simulating laser source. The numerical results of the ray trajectories agree with the visualization results. In addition, the signal of returned ray energy forms sharpen peak just before the sensing edge bursting gas-liquid surface, same as experiment.

10:56AM HJ.00003 Deformation and Magnetophoresis of Bubbles in Magnetic Fluids (Ferrofluids) , PHILIP YECKO, Montclair State University, WAH-KEAT LEE, Argonne National Laboratory, RUBEN SCARDOVELLI, Universita di Bologna, A. DAVID TRUBATCH, Montclair State University — The deformation and coalescence of small bubbles in magnetic fluid (ferrofluid) are directly observed and measured using a novel X-ray phase contrast imaging technique. High resolution X-ray images and videos of water-based ferrofluid (EMG-607/707) reveal: (i) vapor bubbles having diameters in the range 50 to 1000 microns form readily, (ii) neighboring bubbles chain together readily, but do not merge, and (iii) the dynamics of interfaces (bubbles) in ferrofluids are modified by the presence of long chain-like aggregates of the constituent magnetic nanoparticles. Isolated bubbles larger than a few hundred microns become visibly prolate in an applied field, elongating along the field direction. The observed deformations compare favorably with theoretical estimates and the results of direct numerical simulations (DNS). Coalescence is driven by attractive magnetophoretic force induced by non-uniform fields associated with individual bubbles, as was verified by comparing coalescence dynamics to theoretical prediction and to detailed results from DNS. Chain like aggregates of magnetic nanoparticles aligned with the applied field were ubiquitous and led to anisotropic field dependent drag effects. A multiple color function Volume of Fluid (VOF) technique was used to suppress bubble merger in DNS.

11:09AM HJ.00004 Dynamics of Bubble Rising in Vertical and Inclined Square Channel , LUZ AMAYA-BOWER, City College and Graduate Center of the City University of New York, TAEHUN LEE, City College of the City University of New York — A stable Lattice Boltzmann Equation (LBE) Model based on the Cahn-Hilliard diffuse interface approach is used to investigate the dynamics of a bubble rising in a vertical and inclined square channel with large density and viscosity ratios. Deformation parameter (Δ) and terminal velocity (U_t) of the bubble are interrelated quantities which depend on non-dimensional numbers such as Bond Number (Bo), Morton Number (Mo) and ratio between bubble diameter and channel width (κ). This study confirms the relationship between κ and Δ and film thickness (δ), as it was reported by previous experimental studies. As κ is increased, higher Δ and smaller δ are exhibited. This finding is independent of the value of Bo and Mo. In addition, an evaluation was performed for inclined channel to relate the non-dimensional value Froude Number (Fr) and the inclination angle (θ) as function of Bo and Mo. For each set of values of Bo and Mo, there is a critical value of θ which corresponds to the highest value of Fr, consequently highest U_t. This finding is consistent previous simulation and experimental results. This study was performed using a range of Bo and Mo, ($10^{-5} < \text{Mo} < 10^2$) and ($1 < \text{Bo} < 30$), and the inclination of the channel is varied from 0° to 75°.

11:22AM HJ.00005 Microcantilevers with strain gauges as active and passive bubble sensors , MATTHEW STEGMEIR, ELLEN LONGMIRE, Aerospace Engineering & Mechanics, University of Minnesota, SUSAN MANTELL, MUBASSAR ALI, Mechanical Engineering, University of Minnesota — In the current study, we investigate the sensitivity of microfabricated cantilevers with integrated strain gauge sensing to collisions of air bubbles in confined water channel flow. The vertical channel used is 585mm long with a 10mmx2mm cross-section. Flow is upward. Bubbles of diameter 400-2000μm are examined. Flow Reynolds numbers based on mean fluid velocity and hydraulic diameter of 1000-2500 are considered. Cantilevers extend from the center of the 2mm wall and are oscillated perpendicular to the flow direction. Bubbles are introduced upstream of cantilever mounting location and travel in the direction of the flow. Cantilever sizes of 6mmx2mm and 3mmx0.5mm with thickness $\sim$125μm and resonance frequencies of 340Hz and 2670Hz in water are considered. Bubble impacts are recorded using a high frame-rate camera. Strain gauge data are correlated with images. Active and passive results are considered for each beam. Changes in the instantaneous sinusoidal amplitude of the strain gauge signal are used to detect impacts. The effects of impacts on the signal from the strain gauge will be discussed. Results indicate that active sensing using the shorter, stiffer beam is most effective at detecting bubble impacts. Supported by the National Science Foundation (CMS-0300125).

11:35AM HJ.00006 Transient Pressure Measurements in Microscale Bubble Flows , SIAVASH ASLAN-BEIGI, Department of Physics, University of Toronto, MICHAEL MCNEIL, LIAN LENG, AXEL GUENTHER, Department of Mechanical and Industrial Engineering, University of Toronto — Microscale bubble flows have found a wide range of applications in lab-on-a-chip systems and for microchemical synthesis. Most previous studies of these flows have relied on optical micrographs of the fluid phase distribution. We present time-resolved measurements of the fluid pressure during steady flow along the channel that provide for a very sensitive tool to discriminate between flow regimes, to determine bubble/droplet velocities and – in multichannel arrangements - to achieve flow synchronization. We integrated piezoresistive pressure transducers in soft lithographically patterned microfluidic devices. On-chip integration significantly reduced the available dead volume and removed any unwanted bubbles in the sensing channel. The integrated pressure sensors were calibrated and electrically amplified. Dynamic pressure measurements at millisecond time resolution and a sensitivity exceeding the capillary pressure by at least one order of magnitude were obtained. The presented results were obtained at different wetting conditions and for segmented gas-liquid flows at Capillary numbers of 0.001-0.05.

11:48AM HJ.00007 An Experimental Investigation of the Implosion of Cylindrical Shell Structures[1] , C.M. IKEDA, J. WILKERLING, J.H. DUNCAN, University of Maryland — An experimental study of the physics of the implosion of cylindrical shell structures in a high-pressure water environment was performed. The shell structures are filled with air at atmospheric pressure and the implosions occur when the water pressure is raised above the shell buckling stability limit. High-speed photography (27,000 fps) was used to observe and measure the motion of the structure during its implosion. High-frequency underwater blast sensors recorded dynamic pressure waves at 13 positions in the tank. The cylindrical models are made from various aluminum alloys (diameter D = 39.1 mm, wall thickness t = 0.89 mm) and brass (D = 16.7 to 25.4 mm, t = 0.33 to 0.36 mm). The ends of the tubes were sealed with Aluminum caps. The pressure records are interpreted in light of the high-speed movies. Cylinder length-to-diameter (L/D) ratios between 6 and 10 were examined; in this range the cylinders implode in a mode 2 cross-sectional shape at pressures between 6.9 and 28.7 bar. It is found that the pressure versus time records from sensors placed at the same dimensionless radial position (r/D) from the cylinder surface scale well with time and pressure scales from cavitation bubble collapse theory.

[1]This work is supported by the Office of Naval Research, grant number N000140410701.

12:01PM HJ.00008 Bubble bursting mediated aerosols , HENRI LHUISSIER, EMMANUEL VILLERMAUX, Aix-Marseille Universite, IRPHE — Wave breaking over the ocean in the surf zone is responsible for a substantial amount of atmospheric aerosols production. The objects mediating their formation are bubbles entrained below breaking waves, and bursting at the sea surface. We describe the mechanisms by which the liquid shell constitutive of a bubble ultimately results into small drops, also called *film drops*. A bubble bursts when a hole nucleates through the liquid shell. The hole grows at the Culick velocity balancing inertia with surface tension and is bordered by a rim collecting the shell liquid. This initially smooth toroidal rim corrugates when the centripetal acceleration caused by the recession motion is strong enough to trigger a Rayleigh-Taylor destabilization. Ligaments then emerge from corrugations crests and resolve by a Plateau-Rayleigh mechanism into droplets. The final myst properties are thus solely determined by the shell geometry at the bursting onset. It depends on the ratio of the bubble radius to the capillary length, and on the slow gravity drainage of the liquid on which are superimposed rearrangements due to the marginal regeneration at the bubble foot. Our findings will be discussed in connexion with know facts in that context.

12:14PM HJ.00009 Faraday waves at foam/liquid interface[1] , HERVE CAPS, GILES DELON, GRASP-Optofluidics — Monodisperse foams are produced in Hele-Shaw cells and submited to vertical oscillations. This leads to a destabilization of the interface bewteen the foam and the liquid bath. Faraday instability has been identified and characterized in terms of wavelength and destabilization threshold. The control parameters which are considered are: the frequencies of the oscillations, varying between 10 Hz and 100 Hz, the amplitudes of the oscillations, ranging from 1 mm to 15 mm and the bubble size, which is millimetric. Considering the foam as a continuous media, the waves at the interface can be linked to an effective surface tension of the interface, wich depends on the bubble size and the oscillation parameters. Results are interpreted with help of fast-cam recording and energy considerations. This study paves the way to a new continuous approach of liquid/foam interfaces.

[1]GD thanks SSTC-Prodex program for financial support.

12:27PM HJ.00010 Forced Aspiration of bubbles into a capillary tube , MELANIE DURTH, University of Seville, Spain, CHRISTOPHE CLANET, Ecole Polytechnique, France, JUAN FERNANDEZ, University of Seville, Spain — One way to remove lodged bubbles in small vena is to force the bubble to be completely aspirated into a fine needle. We study the aspiration of a bubble into a vertical capillary tube, for different bubble size relative to the capillary diameter (i.e. bubble confinement) and low Bond numbers (pipette diameter $<<$ capillary length). In this case, there is a critical condition of flow rate depending on the bubble confinement and the capillary number Ca beyond which the bubble is aspirated completely into the capillary. Below this value, the bubble breaks-up forming a liquid slug at the entrance of the tube. A simple model which takes into account the draining time of the annular liquid thin film and the characteristic time of the capillary instability, explains the observed experimental results and establish the characteristic time to aspirate completely the bubble.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HK Multiphase Flows IV 101J

10:30AM HK.00001 ABSTRACT WITHDRAWN –

10:43AM HK.00002 Molecular Gas Dynamics on Evaporation and Condensation Induced by Nonlinear Gas Oscillation , MASASHI INABA, Division of Mechanical and Space Engineering, Hokkaido University, TAKERU YANO, Department of Mechanical Engineering, Osaka University, MASAO WATANABE, SHIGEO FUJIKAWA, Division of Mechanical and Space Engineering, Hokkaido University — The resonant gas oscillation excited in a finite one-dimensional space between an oscillating plate (sound source) and a vapor-liquid interface is studied. In particular, we focus on the case where the vapor is a polyatomic gas, e.g. water vapor, the liquid is its own condensed phase and no other species of molecules are existent in the space. The behavior of the gas accompanied with the evaporation and condensation at the interface is analyzed by applying the asymptotic theory for Kn $\ll$ M $\ll$ 1 to the polyatomic version of Gaussian-BGK Boltzmann equation, where Kn is the Knudsen number defined by the ratio of the mean free path of gas molecules and M is the Mach number defined by the ratio of the maximum speed of oscillating plate to the sound speed. The result shows that the gas region consists of the three regions of that governed by wave equation, the thermal boundary layer and the Knudsen layer.

10:56AM HK.00003 One-Dimensional Steady Solutions and Correlations of Internal Condensing Flows in Channels/Tubes and their Comparisons with Experiments , SOUMYA MITRA, AMITABH NARAIN, SHANTANU KULKARNI, Michigan Technological University — Quasi-1 D steady simulation for internal condensing flows that employs solutions of singular non-linear ordinary differential equations is presented along with the governing equations and computational approach. The computational simulation results presented are for internal condensing flows in channel and tube geometry. The quasi-1 D steady simulation results are compared to the experimental results as well as full 2-dimensional CFD based results. These results are shown to be self-consistent and in agreement with one another. The paper further reports some reliable and useful correlations for internal condensing flows (covering most refrigerants of common interest) for the gravity driven case. The quasi-1 D steady simulation results are available from micrometer to larger scale condensers and in various gravity environments. For the micro-meter scale condensers, a critical diameter condition is identified, below which the flows are insensitive to the orientation of the gravity vector as the condensate is always shear driven. The paper discusses transition from gravity dominated flow to shear dominated flows. Paper outlines the difference between transverse gravity and zero gravity flow in a channel.

11:09AM HK.00004 An Introduction to mm Scale Low Kinematic Viscosity Horizontal Channel Flow Morphology in Condensation and Pool Condensation in the Presence of Vapor Flow[1] , MICHAEL KIVISALU, AMITABH NARAIN[2], JORGE KURITA, Michigan Technological University — Preliminary experimental flow regime types have been observed for partial and full condensation in a horizontal channel of millimeter scale. Effects of hydrostatic pooling, wall vibrations, vapor flow pulsations, and solid-liquid-vapor contact on the top surface of the condenser are observed. Wave phenomena such as wave reflection, standing waves, and waves on the interface of vapor bubbles are presented in 2 and 3 dimensions. Due to the high density and low kinematic viscosity of the liquid phase of the fluid (perfluorohexane), the effects observed are primarily inertial. It is found that in these flows pressure pulsation and surface tension effects very strongly influence the morphology of the interface where it contacts the top wall. Wall vibrations contribute to standing waves on the interface. This investigation outlines some of the types of flow one might expect to encounter in a millimeter scale horizontal condenser with wall vibration and pressure pulsation where inertial effects dominate the liquid flow.

[1]Experimental Hardware Courtesy of NASA Grant NNC04GB52G

[2]Faculty Member

11:22AM HK.00005 Experimental study on condensation heat transfer inside a single vertical tube[1] , JORGE KURITA, AMITABH NARAIN, MICHAEL KIVISALU, Michigan Technological University — In this paper, results from a new computational model for condensation inside vertical cylindrical tubes have been compared to test data for FC-72 obtained in an actively controlled flow loop. This correlation was developed taking in account thermo physical properties of the 9 most common refrigerants. The present paper reports heat transfer coefficient, length of full condensation, vapor quality and direct measurement of the saturation and wall temperatures during condensation within a single cylindrical tube of 6.6 mm diameter and 700 mm length. The new model has been tested over the following range of experimental conditions: mass flux from 2.9 to 87.7 kg/(m2s), vapor qualities from 0.33 to 0.64 and full condensation. The saturation minus wall temperature difference was varied from 5 to 45 C. This model predicts heat transfer coefficients to within 15 percent accuracy over 70 percent of data points and predicts within 30 percent accuracy over 87 percent of data points in full and partial condensation cases respectively.

[1]NSF Grant No. CTS-0086988 and NASA Grant No. NNC04GB52G.

11:35AM HK.00006 Condensing Flows in High Aspect Ratio Channel Geometries[1] , CATHERINE KOVEAL, MATTHEW MCCARTHY, EVELYN N. WANG, Massachusetts Institute of Technology — We investigate condensing flow regimes in high aspect ratio rectangular geometries, where the width to height ratio ranges from 20-to-1 to 65-to-1. These geometries are designed for condenser layers of a novel capillary-pumped loop heat pipe design in a high performance heat sink. In this work, we study the effect of geometry, vapor mass flow rate, and surface design on condensing flow regimes and heat removal capability. We fabricated an experimental test rig which allows for optical access from the top, temperature measurements on the condensing surface, and controlled cooling from the bottom. In addition, the rig was placed on a tilt stage to examine the effect of a gravitational head on condensation. The experimental results show that the flow regimes are largely dictated by the dominant force, i.e., gravity, surface tension, or inertia. In addition, as the backside cooling temperature increased, the condensing length increased. Current work is focused on incorporating surface features to enhance heat transfer coefficients and to eliminate unstable condensation regimes.

[1]We gratefully acknowledge our funding support from the DARPA MACE program with Dr. Tom Kenny as program manager.

11:48AM HK.00007 Collisions in a liquid fluidized bed , ALICIA AGUILAR-CORONA, Laboratoire de Genie Chimique, Toulouse France, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico, OLIVIER MASBERNAT, Laboratoire de Genie Chimique, Toulouse France — Collisional phenomena in a liquid fluidized bed were studied in terms of two parameters: the collision frequency and the coefficient of restitution. Experimental measurements of these parameters were conducted by particle tracking in an index-matched array. Collision detection was based on the use of a peak acceleration threshold of the instantaneous speed of dark tracers. The measurements of collision frequency were compared with the theoretical expression derived from the kinetic theory for granular flow (KTGF). The normal and tangential restitution coefficients were measured from the trajectories before and after contact for both particle-particle and particle-wall collisions. A comparison with previous theoretical and experimental works is presented and discussed.

12:01PM HK.00008 Multi-dimensional Shock Fluidization of Particle Assemblies , SVETLANA SUSHCHIKH, VLADIMIR MITKIN, CHIH-HAO CHANG, THEO THEOFANOUS, UCSB — We present new kinds of experiments that resolve the transient dynamics of shock-fluidized particle (glass, lead) assemblies in 1D and 2D geometries, covering both subsonic and supersonic after-shock flows (shock Mach numbers 1.5 to 2.5). From high speed video records time-wise evolutions of particle number densities could be recovered, as on this basis complete clouds (with spatially-distributed particle number densities) could be reconstructed at various positions along the flow channel, up to 1.3 m away from the point of initial interaction. We also discuss numerical simulations, based on a two-fluid model made consistently hyperbolic over the complete range of Mach numbers, thus allowing the grid-refinements needed to capture sharp density gradients at such intense flow conditions. These simulations are a priory in that the only constitutive law is for the drag coefficient; it is obtained from single- particle tests and direct numerical simulations, augmented with standard approaches to account for particle proximity effects. These simulations are in excellent agreement with the experiments, and so are shown to be sample, initial results of overall cloud dynamics in explosive dissemination of bulk liquids (10's of kilogram quantities, km/s speeds).

12:14PM HK.00009 Crack propagation speed in quasi two-dimensional dry foam[1] , SHEHLA ARIF, Mechanical Engineering, Northwestern University, SASCHA HILGENFELDT, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Quasi two-dimensional dry foam in a Hele-Shaw cell is used as a model system to study crack propagation, with the mm-sized bubbles playing the role of discrete atoms or molecules in a lattice. Pressurized air induces finger-like ductile failure (at low rate of applied stress) or brittle cleavage with breakage of successive films (at high rate of applied stress). We find that the upper limit speed of ductile cracks is lower than the lower limit speed of brittle failure (a velocity gap exists). To understand both processes requires balancing viscous and surface tension forces in the Plateau borders of the foam bubbles. Time scales can then be derived for the lateral gliding of bubbles past each other (necessary for ductile failure) and the stretching and rupture of films (necessary for brittle failure). We compare the derived brittle-crack speeds with the experimentally observed values of several m/s, resolved by high-speed photography.

[1]We acknowledge support from the American Chemical Society Petroleum Research Fund under ACS PRF # 46185-AC5.

12:27PM HK.00010 Direct Current Electrorheological Stability Determination of Water-in-Crude Oil Emulsions , VLADIMIR ALVARADO, XIUYU WANG, University of Wyoming — Emulsion stability is a fundamental determination for separation technologies. We use the critical electric field (CEF) and viscosity changes in DC eletrorheological (ER) experiments in dynamic mode to determine the stability of water-in-crude oil emulsions, previously studied through bottle tests. The CEF value corresponds to the value of electric field at which the current reaches 95% or larger of the plateau value. The results show that CEF can be consistently obtained through current measurements, resulting from emulsion structure breakdown. Viscosity changes are not good proxies of stability unless a robust emulsion structure is found. Emulsion structure breakdown is explored through rheological characterization before and after voltage sweeps have been performed. When the electric field applied is below the CEF value, the storage and loss moduli responses as well as viscosity as functions of frequency are recovered. However, when the electric field is greater than the CEF value, the emulsion structure breaks down irreversibly.

Monday, November 23, 2009 10:30AM - 12:27PM –
Session HL Acoustics I 200A

10:30AM HL.00001 Scattering of an entropy disturbance into sound by a symmetric thin body[1]

, DANIEL BODONY, University of Illinois at Urbana-Champaign — The interaction of a convecting entropy disturbance, such as generated by a gas turbine combustor, with a solid object is known to generate sound. The sound generation is due to (i) the acceleration of the convected disturbance by the mean flow and (ii) satisfaction of the wall-boundary condition on the object. This process, which leads to the so-called indirect combustion noise, is known to be present in modern gas turbine engines but its specific details are not known, including its overall contribution to the acoustic signature of the engine and its influence on the combustor. Computational and analytical results are presented to examine the sound field created by a localized entropy disturbance convecting in the vicinity of a symmetric thin body. Unsteady calculations of the compressible Euler equations are used to directly compute the radiated sound. Rapid distortion theory is used, when combined with a low frequency Green's function, to estimate the dominant radiated field. It is found that the source structure is dominated by a streamwise-oriented dipole, with the most important interactions occurring near the stagnation points.

[1]Supported by the NASA fundamental aeronautics program (subsonic fixed wing), project number NNX07AB90A.

10:43AM HL.00002 Interaction of sound with an elastic plate in a duct[1]

, MAHESH SUCHEENDRAN, DANIEL BODONY, PHILIPPE GEUBELLE, UIUC — The interaction of sound with a cavity backed elastic plate in a duct is studied using analytical and numerical methods. The problem consists of an incoming plane wave in an infinite duct with a finite elastic plate mounted flush on one of the walls of the duct. The analytic solution is found by series expansion of pressure in the cavity, duct and velocity of the plate using cosine and sine modes that satisfy the boundary conditions. Two separate boundary conditions for the plate are considered, simply-supported and clamped. The response of the plate is quantified by V_{rms}, the square root of integral average of velocity square of plate. A corresponding numerical simulation, based on first principles, of the same problem further illustrates the sound-plate interaction. Variation of V_{rms} of the plate with forcing frequency shows that the acoustic-structure interaction modifies the frequency of peak response from that of the natural frequency of the plate.

[1]The authors gratefully acknowledges the financial support of the Midwest Structural Sciences Center (MSSC). MSSC is supported by the U.S. Air Force Research Laboratory Air Vehicles Directorate under contract number FA8650-06-2-3620.

10:56AM HL.00003 ABSTRACT WITHDRAWN –

11:09AM HL.00004 The sound of boundary-layer flow over a roughness patch[1]

, QIN YANG, MENG WANG, University of Notre Dame — The sound radiation by a turbulent boundary layer over an array of 4 (spanwise) by 10 (streamwise) hemispherical roughness elements is studied using large-eddy simulation and Lighthill's theory. The roughness height is 12.7% of the boundary layer thickness and 17% of the spacing between neighboring elements in both directions. The momentum-thickness based Reynolds number is 2984. The acoustically compact roughness elements and their images in the wall radiate primarily as acoustic dipoles in the plane of the wall. Similar to previous findings with a pair of roughness elements, the dipole sources are mainly generated by the interaction of roughness elements with incoming turbulent eddies and horseshoe vortices. Spanwise dipoles are stronger than streamwise dipoles in the low and intermediate frequency range, and wake turbulence enhances sound radiation from downstream elements. It is found that the leading row of roughness elements produces the weakest sound. After a small overshoot by the second row, the rows further downstream generate sound of comparable intensity, which is stronger than that of the first row. The correlations between dipole sources associated with neighboring roughness elements are weak, suggesting that the roughness elements radiate as essentially independent sources.

[1]Supported by ONR Grant N00014-06-1-0640.

11:22AM HL.00005 Acoustic source mechanisms for boundary-layer flow over small steps[1]

, MENG WANG, MINSUK JI, University of Notre Dame — The aeroacoustics of low-Mach-number boundary-layer flow over small backward and forward facing steps is studied using large-eddy simulation and Lighthill's theory with a low frequency (compact step height) Green's function. The Reynolds number based on the step height and free-stream velocity ranges from 328 to 21000 as the step height varies from 0.83% to 53% of the boundary layer thickness. The steps act primarily as acoustic dipole sources aligned in the streamwise direction. Consistent with previous experimental measurements, the forward step is louder than the backward step, because it generates stronger sources in regions closer to the step corner, which is heavily weighted by the Green's function. A detailed analysis of flow field and Green's function weighted sources reveals that the backward step generates sound mainly through diffraction of the boundary-layer source field which is not much affected by the step in the acoustically important region, whereas the forward step generates sound through a combination of diffraction and turbulence modification by the step. As the step height decreases, the difference in sound level between forward and backward steps is much reduced as turbulence modification becomes less significant.

[1]Supported by ONR Grant N00014-09-1-0602.

11:35AM HL.00006 Accurate calculation of high-frequency sound generated by interaction of low Mach number flows and rigid bodies[1]

, YASER KHALIGHI, PARVIZ MOIN, Stanford University — According to N. Curle, the interaction of turbulent flow and rigid bodies is an efficient mechanism of sound generation at low Mach number regimes. A popular approach to compute the sound due to this interaction is to use an approximation of Curle's solution to Lighthill's equation. In this approximation, pressure on rigid surfaces is replaced by the hydrodynamic pressure which can be easily obtained from the solution of incompressible Navier-Stokes equation; however, this approximation is known to be valid only at low frequencies and under-predict the sound at high frequencies. The objective of the present study is to improve this approximation for the high frequency range. In this work we construct the high frequency sound by formally decomposing the surface pressure into contributions from hydrodynamics and acoustics. The acoustic pressure on the surface is obtained by splitting the acoustic Green's function and solving a boundary integral equation. The projection of surface acoustic pressure to the farfield compensates for the missing portion of sound at high frequencies. This method is applied to the problem of sound generated by turbulent vortex shedding of a cylinder at Re=10,000.

[1]This work is supported by Franklin P. and Caroline M. Johnson Fellowship.

11:48AM HL.00007 Acoustic chambers for sonofusion experiments - sensitivity on geometry and materials

, MARKUS J. STOKMAIER, Karlsruhe Institute of Technology, Germany, RICHARD T. LAHEY JR., Rensselaer Polytechnic Institute, NY, ANDREAS G. CLASS, Karlsruhe Institute of Technology, Germany, BERNARD A. MALOUIN, Rensselaer Polytechnic Institute, NY, THOMAS SCHULENBERG, Karlsruhe Institute of Technology, Germany — Sonofusion (SF) relies on the perfect realization of a symmetrically imploding vapor bubble in a compressing acoustic field, so that the center of the singularity yields extreme energy densities, potentially allowing for thermonuclear fusion to occur. An assembly of a cylindrical acoustic chamber with longitudinal wave reflectors allows excitation of the fundamental mode of the liquid body. Our finite-element simulations of such a vibrating glass chamber filled with liquid attempt to answer questions that have arisen since the claimed successful SF experiments of Taleyarkhan. In particular, we show that the sensitivity to geometry and materials of the acoustic chamber may be the reason why SF is apparently difficult to reproduce. This could be a reason why completely independent confirmation of SF is still lacking. A 2-D axisymmetric forced harmonic analysis in ANSYS, is presented and compared to own measurements of pressure amplitude and wall displacement.

12:01PM HL.00008 Molecular simulation of sound propagation , TAKERU YANO, Osaka University — Numerical simulation of molecular dynamics of sound propagation in a gas is carried out to clarify the propagation property of sound waves with large amplitude and very high frequency. Assuming the Lennard–Jones inter-molecular potential, we calculate the motions of hundreds of thousand of monatomic molecules excited by an oscillating plate in a gas phase. The result is compared with corresponding numerical solutions of model Boltzmann equation and Navier–Stokes equations.

12:14PM HL.00009 A Formulation of Compressing Process and Aeroacoustics , JIE-ZHI WU[1], FENG MAO, LIJUN XUAN, WEIDONG SU, YIPENG SHI, Peking University, Beijing, China — We study the general theoretical formulation of compressing (longitudinal) process and aeroacoustics. A maximum Helmholtz decomposition (MaxHD) principle is proposed as the basic criterion of the formulation, which is applied at two levels. At the fundamental level, the MaxHD leads to a convective wave equation for dilatation as the counterpart of the diffusion equation for vorticity in the shearing (transverse) process. This is the fundamental governing equation for the compressing, equivalent to, among others, Lilley's (1973) 3rd-order equation for logarithmic pressure and, for inviscid flow, Howe's (1975) total-enthalpy equation. Phillips' (1960) 2nd-order equation is disqualified by MaxHD. At the operational level, the MaxHD further isolates the compressing variables from the others by decomposing the velocity itself. This yields a 4th-order convective wave equation for the velocity potential. Unlike equations of Lilley, Howe, and other equivalents, the source terms do not contain compressible velocity potential, and the vortical velocity appears only in source terms and operator coefficients. Thus, its linearized version should be the rational basis for problems of sound wave in various shear flows.

[1]Also at University of Tennessee Space Institute, Tullahoma, TN 37388, USA

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HM Aerodynamics 200B

10:30AM HM.00001 Span efficiency of wings at moderate Reynolds number , GEOFFREY SPEDDING, University of Southern California, JOHN MCARTHUR, Aerovironment Inc. — Classical aerodynamic models of lift and induced drag of finite wings work well for chord-based Reynolds numbers of 10^6 or more, where the thin, viscous boundary layers are mostly attached. Recent interest in the design of practical micro-air vehicles and in their natural counterparts (birds, bats, larger insects) brings us into a Reynolds number regime (10^4 – 10^5) where the aerodynamic performance is strongly affected by the possibility of separation of the laminar boundary layer, and then its possible re-attachment. Now, corrections to the inviscid formulations of induced drag become substantial and difficult to estimate. Inconsistencies in the literature further confuse the issue. Here, we clarify the definitions of span efficiency and test the applicability of one-parameter correction models for moderate Reynolds numbers. Suggestions for model improvements will follow.

10:43AM HM.00002 On the initial-value problem for the flow past a finite wing or blade , JOHN RUSSELL, Fla. Inst. of Tech. — Assume piecewise irrotational motion of an incompressible fluid in which the normal velocity distribution on the wing is given and the wake is subject to continuity of normal velocity and pressure across it. The boundary integral for the velocity potential ϕ outside the wing simplifies when its inside is filled with fluid whose motion satisfies similar field equations and whose normal velocity is continuous with that outside. Calculation of $\mathbf{u} = \nabla\phi$ for the velocity transforms a superposition of dipoles to a superposition of quadrupoles and thus to integrals that diverge in the limit as the field point approaches the boundary. Integration by parts via STOKES theorem yields an integral for the velocity of BIOT-SAVART type (i.e. involving dipoles). Equations for the evolution of the vortex sheet follow from equations in §2.2 of SAFFMAN, P.G. *Vortex Dynamics*, Cambridge, 1992 and admit interpretations analogous to those of HEMHOLTZ vortex theorems but applied to vortex ribbons instead of vortex tubes.

10:56AM HM.00003 Investigation of a low Reynolds number airfoil using molecular tagging velocimetry[1] , ALAN KATZ, AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — Molecular tagging velocimetry (MTV) is used to study flow separation, transition, and reattachment on a SD7003 airfoil at a chord Reynolds number of 20,000. Multi-line tagging is employed to obtain high resolution one-component velocity data near the suction surface of the airfoil. The spatial resolution in the wall-normal direction normalized by the chord length is about 0.0003, nearly ten times smaller than previously reported PIV measurements on the same airfoil under similar flow conditions. Preliminary data will be presented for the locations of flow separation and reattachment for the airfoil at an angle of attack of 8 degrees, and results will be compared with existing experimental and computational data.

[1]This work was supported by AFRL through the Michigan/AFRL Collaborative center in Aeronautical Sciences.

11:09AM HM.00004 Lift production through asymmetric flapping , SHREYAS JALIKOP, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, K.R. SREENIVAS, Jawaharlal Nehru Centre for Advanced Scientific Research — At present, there is a strong interest in developing Micro Air Vehicles (MAV) for applications like disaster management and aerial surveys. At these small length scales, the flight of insects and small birds suggests that unsteady aerodynamics of flapping wings can offer many advantages over fixed wing flight, such as hovering-flight, high maneuverability and high lift at large angles of attack. Various lift generating mechanims such as delayed stall, wake capture and wing rotation contribute towards our understanding of insect flight. We address the effect of *asymmetric* flapping of wings on lift production. By visualising the flow around a pair of rectangular wings flapping in a water tank and numerically computing the flow using a discrete vortex method, we demonstrate that net lift can be produced by introducing an asymmetry in the upstroke-to-downstroke velocity profile of the flapping wings. The competition between generation of upstroke and downstroke tip vortices appears to hold the key to understanding this lift generation mechanism.

11:22AM HM.00005 Unsteady Vortex Structures in the Wake of a Piezoelectric Flapping Wing[1] , LUCAS CLEMONS, HIROFUMI IGARASHI, HUI HU, Aerospace Engineering Dept., Iowa State University — An experimental study was conducted to characterize the behavior of Unsteady Vortex Structures in the Wake of a piezoelectric flapping wing with miniaturized size (about 10mm in chord length), large flapping amplitude (up to 2.0 times of chord length) and high flapping frequency (60Hz) to explore the potential application of piezofans as the compact, gearless flapping-wings for the development of novel piezoelectric-flapping-wing-based Nano-Air-Vehicles (NAVs). The experimental investigation was performed in a low-speed wind tunnel. A digital particle image velocimetry (PIV) system was used to achieve phased-locked flow field measurements to quantify the transient behavior of the unsteady vortex structures in wake of the piezoelectric flapping wing. The effects of important parameters such as incoming flow velocity (i.e., forward flight speed), the flapping amplitude, and the incline angle of the flapping wing in relation to the incoming flow direction (i.e. the angle of attack) on the wake vortex shedding processes were examined to elucidate underlying physics in order to explore/optimize design paradigms for the development of novel piezoelectric-flapping-wing-based NAVs.

[1]Supported by NSF-CAREER Program, Award No. CTS-0545918.

11:35AM HM.00006 The Effect of Flexible Membrane Scalloping on the Lift and Drag of Flat Plates in Low Re Flow[1] , MELISSA CONWAY, JAMES HUBNER, University of Alabama — Micro Air Vehicles (MAVs) have the potential to be used for surveillance and assessment of dangerous environments. Promising applications for MAVs encourage researchers to find effective designs for the most adept flying. Previous wake analysis research conducted at the University of Alabama suggested trailing-edge scalloping of the membrane reduces drag. The present study, drawing from the previous research, tests various geometries involving scalloped and nonscalloped trailing-edges on a three-component force balance. Flat plates were tested to mimic fixed MAV wings at low Reynolds number flow and varying angles-of-attack in a low-speed wind tunnel. Thin aluminum plates with repeated cell geometries were covered with a flexible latex membrane. Cell aspect ratios of 1 and 2 (80% and 40%, respectively, of the chordwise length) were tested along with a solid flat plate and a solid scalloped plate. The presentation will discuss results and explore the impact of changing geometries on lift, drag and aerodynamic efficiency.

[1]Work performed under REU site, NSF grant EEC 0754117.

11:48AM HM.00007 Flow-induced oscillation of free reeds , PETER BUCHAK, JOHN BUSH, MIT — We present an investigation of the mechanism by which air flow induces oscillation in free reeds, which produce sound in several musical instruments. In this system, a thin strip of metal is clamped at one end to the top of a plate above a slot large enough for it to pass through. This geometry allows a uniform flow to induce and sustain large-amplitude oscillations of the reed. We investigate experimentally the conditions under which oscillation occurs and formulate a theoretical model, informed by the work of St. Hilaire et al. [JFM, 1971], to explain our observations.

12:01PM HM.00008 The Roberto Carlos Spiral , CHRISTOPHE CLANET, Ecole Polytechnique, France, GUILLAUME DUPEUX, ANNE LE GOFF, Ecole Polytechnique, DAVID QUERE, ESPCI — We study the motion of spinning spheres in a fluid at high Reynolds number and show that, in the "low gravity limit," their trajectory is a spiral, the curvature of which increases exponentially. We show that this spiral was used by Roberto Carlos in soccer to score amazing goals and then discuss its importance in other ball games.

12:14PM HM.00009 A Study of the Effects of Large Scale Gust Generation in a Small Scale Atmospheric Wind Tunnel: Application to Micro Aerial Vehicles , JASON ROADMAN, KAMRAN MOHSENI — Modern technology operating in the atmospheric boundary layer could benefit from more accurate wind tunnel testing. While scaled atmospheric boundary layer tunnels have been well developed, tunnels replicating portions of the turbulence of the atmospheric boundary layer at full scale are a comparatively new concept. Testing at full-scale Reynolds numbers with full-scale turbulence in an "atmospheric wind tunnel" is sought. Many programs could utilize such a tool including that of Micro Aerial Vehicles (MAVs) and other unmanned aircraft, the wind energy industry, fuel efficient vehicles, and the study of bird and insect fight. The construction of an active "gust generator" for a new atmospheric tunnel is reviewed and the turbulence it generates is measured utilizing single and cross hot wires. Results from this grid are compared to atmospheric turbulence and it is shown that various gust strengths can be produced corresponding to days ranging from calm to quite gusty. An initial test is performed in the atmospheric wind tunnel whereby the effects of various turbulence conditions on transition and separation on the upper surface of a MAV wing is investigated using oil flow visualization.

12:27PM HM.00010 Spiral motion of free falling disks , HONGJIE ZHONG, CUNBIAO LEE, LTCS and College of Engineering, Peking University — We report on experimental work on circular thin disks falling freely in quiescent water. A stereoscopic vision method was applied to measure the evolution of the six degrees of freedom. By analyzing the digital images of the disk obtained by two fixed CCD cameras, the position and orientation of the body were measured throughtout the fall. Translatory and angular motions,as well as the force and torque induced by the flow were determined. A new helical motion type was observed for moment of inertia $I^* < 5 \times 10^{-4}$ and Reynolds number in the range of $500 - 3000$. This type of helical motion is distinct from the one caused by eccentricity. For the eccentricity case, the angular motion of the disk is purely precession whereas gyration and procession were both present for the low I^* case. The wake structure of the zigzag motion and spiral motion were visualized in combine with PIV measurement. The instability mechanism of the wake structure, which cause the zigzag to spiral transition, is discussed.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HN Non-Newtonian Flows II 200C

10:30AM HN.00001 Topological fluid mechanics of stirring in a non-Newtonian fluid , XUEMEI CHEN, ANTHONY NELSON, MATTHEW BOKULIC, KELLEN SHAIN, MARK STREMLER, Virginia Tech — It is well known that laminar flows, both Newtonian and non-Newtonian, can be stirred effectively using chaotic advection, which produces exponential stretching and folding of material lines and surfaces. A recent development in this field is the concept of topological choas, in which the topology of the trajectories of moving rods can be used to predict a lower bound on the stretching of the surrounding fluid. We present the first analysis of topological chaos in a non-Newtonian fluid using experimental and computational tools. The power of this approach lies in the fact that a lower bound on the stretching is established independent of the fluid properties, making it possible to predict efficient stirring even when the fluid properties are quite complex and/or not well characterized.

10:43AM HN.00002 Visualization and microrheology of complex fluid/fluid interfaces , SIYOUNG Q. CHOI, JOSEPH ZASADZINSKI, TODD SQUIRES, UCSB Chemical Engineering — We describe a novel microrheological technique to measure the rheological properties of fluid/fluid interfaces, which can dramatically affect the flow properties and dynamics of multiphase materials (emulsions, foams, cells and organs). Such measurements can be particularly challenging, as one needs to measure the influence of molecularly thin, two-dimensional layers but be insensitive to the three-dimensional bulk fluids on either side. However, dimensionality helps here: interfacial forces on a probe are exerted along a contact perimeter, whereas the bulk forces are exerted on the contact area. Smaller probes thus increase the perimeter/area ratio, and therefore the relative sensitivity to interfacial viscoelasticity. We fabricate micron-scale ferromagnetic amphiphilic disks (with versatile surface chemistry), place them on the interface, use external electromagnets to exert a known torque (stress), and measure the resulting rotational displacement (strain). In addition to its sensitivity, our technique can measure frequency dependent linear/nonlinear viscoelastic properties and yield stresses. Simultaneous visualization of the interface by fluorescence microscopy allows us to correlate local dynamics withe measured rheology. We validate our technique and highlight its capabilities with measurements on a variety of systems, including two-dimensional colloidal monolayers, fatty acid and phospholipid monolayers.

10:56AM HN.00003 ABSTRACT WITHDRAWN –

11:09AM HN.00004 Formation of beads-on-a-string structures during the pinch-off of viscoelastic filaments , PRADEEP BHAT, SANTOSH APPATHURAI, MICHAEL HARRIS, Purdue University, West Lafayette, IN 47907, MATTEO PASQUALI, Rice University, Houston, TX 77005, GARETH MCKINLEY, Massachusetts Institute of Technology, Cambridge, MA 02139, OSMAN BASARAN, Purdue University, West Lafayette, IN 47907 — Breakup of liquid filaments is omnipresent in nature and technology. When a filament formed by placing a drop of syrup between a thumb and a forefinger is stretched by pulling apart the two fingers, it resembles a thinning cylinder. If the same experiment is repeated with saliva, the filament's morphology close to pinch-off resembles that of beads of several sizes interconnected by slender threads. Although there is general agreement that formation of such beads-on-a-string (BOAS) morphology only occurs for viscoelastic fluids, the mechanism behind this phenomenon remains unclear and controversial. The physics of formation of BOAS structures is probed here by simulation which reveals that viscoelasticity alone does not give rise to a small, satellite bead between two much larger main drops (beads) but that inertia is required for its formation. Viscoelasticity, however, enhances the growth of the satellite bead and delays pinch-off, which leads to a relatively long-lived, stable beaded filament. The new simulations also show the formation of second-generation sub-satellite beads in certain cases, as observed experimentally but not, heretofore, predicted theoretically.

11:22AM HN.00005 On the evolution of the drop-filament corner region during the pinch-off of viscoelastic fluids , SANTOSH APPATHURAI, PRADEEP BHAT, MICHAEL HARRIS, Purdue University, West Lafayette, IN 47907, MATTEO PASQUALI, Rice University, Houston, TX 77005, OSMAN BASARAN, Purdue University, West Lafayette, IN 47907 — Fluid pinch-off is important in applications involving the production of drops, e.g., ink-jet printing and atomization, and in the capillary breakup extensional rheometry (CaBER). A characteristic feature of fluid pinch-off is the formation of drops that are connected to thinning threads. In the pinch-off of viscoelastic fluids, the region that connects the drops to the threads develops into a sharp corner. Recently, Clasen *et al.* [J. Fluid Mech. **556**, 283 (2006)] showed that such a corner evolves self-similarly—a result which can be exploited in estimating accurately the extensional viscosity of fluids from CaBER experiments. However, the agreement between the similarity solution derived by Clasen *et al.* and experiments is only qualitative, and it may be due to their approximation of the dynamics in the corner region by a one-dimensional analysis. The evolution of the drop-filament corner region is elucidated here using theory and both one- and two-dimensional computations. A new similarity solution is obtained which describes better the shape of the liquid-gas interface in the corner region, and the dynamics in the corner region is demonstrated to be two-dimensional.

11:35AM HN.00006 Tunable adhesion using field-activated "smart" fluids , RANDY EWOLDT, Massachusetts Institute of Technology, PIOTR TOURKINE, Ecole Normale Supérieure, GARETH MCKINLEY, A.E. HOSOI, Massachusetts Institute of Technology — We demonstrate experimentally that field-responsive magnetorheological fluids can adhere to non-magnetic substrates. The tunable adhesive performance is measured experimentally with pull-off tests, i.e. probe-tack experiments, in which the external magnetic field and fluid geometry are varied. The adhesive force is predicted by a lubrication model which treats the adhesive as a yield stress fluid with field-dependent and inhomogeneous yield stress (caused by the inhomogeneous external magnetic field). The peak adhesive force, the "work of adhesion" and the mode of failure are all controlled by the field-responsive nature of the magnetorheological fluid forming the adhesive layer.

11:48AM HN.00007 Branching of an electrospinning fiber , RAHUL SAHAY, CHIANG JUAY TEO, YONG TIAN CHEW, National University of Singapore, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia — The phenomenon of branching during the electrospinning of polymeric liquid has been studied using high-speed video imaging. Linear stability analysis of the electrified jet has been performed including non-Newtonian effects. The branching is only observed using very fine needles of diameters of approximately 100 microns. The onset of the branching is associated with the flattening of the jet, which occurs at stronger electric fields than used for regular fiber spinning. The branch emerges out of the jet where it has the highest azimuthal curvature. We have characterized the relationship between inter-branch distances and the operating parameters, such as the applied electric field and the physical properties of the liquid, such as the molecular weight of polymer and the nature of the solvent.

12:01PM HN.00008 Is there an elongational Flow State in Circular Couette Flow? , ANDREAS ZELL, CHRISTIAN WAGNER, Universität des Saarlandes — We investigate the behavior of dilute polymer solutions in a Taylor-Couette cell with independently rotatable cylinders. The focus of our interest concentrates on the examination of the elongation of the solved polymers and their response on the imposed flow. The elongation is imposed to the fluid by a special ratio of the cylinder rotation rates which create a non rotational circular Couette flow. This special flow state is numerically investigated concerning its effectiveness deforming dispersed material in the fluid. Experimental results show, that the flow state holds for Newtonian fluids but that it is changed for polymer solutions due to their power law shear thinning behaviour. In addition to the investigated polymer solutions including different types of polymers, concentrations and solvent viscosities an industrial manufactured emulsion has been tested.

12:14PM HN.00009 Is there a Relationship between the Elongational Viscosity and the First Normal Stress Difference in Polymer Solutions? , STEPHAN GIER, ANDREAS ZELL, Universität des Saarlandes, SALIMA RAFAI, Laboratoire de Spectrometrie Physique, CHRISTIAN WAGNER, Universität des Saarlandes — We investigate polymer solutions in shear and elongational flow. Shear flow is created in a cone-plate-geometry of a commercial rheometer. The capillary thinning of a filament of polymer solution in the Capillary Breakup Extensional Rheometer (CaBER) serves as an elongational flow. We compare the relaxation time and the elongational viscosity measured in the CaBER with the first normal stress difference and the relaxation time from the rheometer measurements. All these four quantities depend on different fluid parameters - the viscosity of the polymer solution, the polymer concentration within the solution, and the molecular weight of the polymers - and on the shear rate (in the shear flow measurements). Nevertheless, we found that the first normal stress coefficient depends quadratically on the CaBER relaxation time. A simple model is presented that explains this relation on a phenomenological level.

12:27PM HN.00010 An experimental investigation on transient change in viscoelasticity in mixing and reaction processes between a water-soluble polymer solution and a metal ion. , MITSUMASA BAN, YUICHIRO NAGATSU, IWATA SHUICHI, YUTAKA TADA, Nagoya Institute of Technology, Japan — We have experimentally investigated mixing and reaction processes between a partially hydrolyzed polyacrylamide (PAM) solution and iron ion. We added a solution including iron ion to a vessel in which the PAM solution was stirred by an impeller at a constant rotational speed. We observed flow behavior in the vessel during and after the addition of the iron ion solution. We found transient flow behavior. After the addition of the iron ion solution, Weissenberg effect started to occur. It should be noted that Weissenberg effect is not observed before the addition in the present experimental condition. When the rotation of the impeller was kept, Weissenberg effect gradually disappeared and a free surface of the solution finally became flat. In order to elucidate mechanism for the observed transient phenomenon, some measurements by means of a rheometer have been performed. Finally, we propose the mechanism associated with interaction between iron ion and the polymer chain.

Monday, November 23, 2009 10:30AM - 12:40PM –

Session HP Instability: Richtmyer-Meshkov 200D

10:30AM HP.00001 PIV Analysis of the Richtmyer-Meshkov Instability for a He/SF_6 Interface , CHRIS WEBER, NICHOLAS HAEHN, JASON OAKLEY, MARK ANDERSON, RICCARDO BONAZZA, University of Wisconsin-Madison — Experiments are presented for the Richtmyer-Meshkov instability of a nearly single mode interface of helium over sulfur hexafluoride. The initial condition is an interface created by oscillating a pair of rectangular pistons to form a nearly 2D sinusoidal standing wave in a vertical shock tube. The incident shock wave ($M = 1.2$) deposits a large amount of vorticity on this high Atwood number ($A = (\rho_2 – \rho_1)/(\rho_2 + \rho_1) = 0.95$) interface resulting in amplitude growth and asymmetrical spike/bubble development. Particle imaging velocimetry (PIV) is accomplished by seeding the top gas with Al_2O_3 particles and acquiring a pair of planar images that are analyzed to obtain the velocity field. Experimentally determined amplitude growth rate, circulation, and energy spectra are compared with models and numerical simulations.

10:43AM HP.00002 Turbulence measurements in reshocked Richtmyer-Meshkov unstable curtains using stereo-PIV/PLIF , B.J. BALAKUMAR, GREGORY ORLICZ, SRIDHAR BALASUBRAMANIAN, PETER HUMPHREYS, CHRIS TOMKINS, KATHY PRESTRIDGE, Los Alamos National Laboratory — Estimates of two important turbulence quantities, the density self-correlation parameter (b) and the generalized Reynolds stress tensor (R_{12}), have been experimentally obtained in Richtmyer-Meshkov unstable fluid layers after reshock using simultaneous PIV/PLIF diagnostics (Balakumar et. al., Phys. Fluids, 2008). We observe a double-peak structure in "b" at 4 times after reshock, with peaks coinciding with the edges of the turbulent structure. Whole-field histograms of the Reynolds stress follow symmetric distributions with equal contributions of positive and negative values, while spatial contour maps clearly show a streamwise asymmetry with large fluctuations preferentially occurring upstream of the centerline. We will also present preliminary stereo-PIV/PLIF measurements in the shock tube to explore the 3D behavior of the flow field before and after reshock. Statistical convergence estimates will be provided to illustrate the difficulties of obtaining true ensemble estimates in RM flows.

10:56AM HP.00003 Mach number effects in shock-driven instabilities , GREG ORLICZ, B.J. BALAKUMAR, SRIDHAR BALASUBRAMANIAN, CHRIS TOMKINS, KATHY PRESTRIDGE, LANL — Experiments are performed to study the effects of incident shock Mach number on the development of a varicose-perturbed, heavy-gas curtain (air-SF6-air). Incident shock strength is varied from Mach 1.2 to 2.0, and the dynamic evolution of the gas curtain is observed using Planar Laser-Induced Fluorescence (PLIF) and Particle Image Velocimetry (PIV). Previous work at the Los Alamos Gas Shock Tube (Orlicz et al. Phys. Fluids 2009), using the PLIF diagnostic to measure the temporal evolution of the density field, has shown that integral width growth rates for the curtain collapse using a scaling based upon the convection velocity of the curtain. However, the instantaneous mixing rate shows differences in mixing at scales smaller than the integral width, indicating that integral width alone is insufficient to describe the Mach effects on mixing.

11:09AM HP.00004 Oblique shock acceleration of cylindrical gaseous interfaces and interaction with boundary layers[1] , EVAN JOHNSON, MARIO CHAVEZ, PETER VOROBIEFF, C. RANDALL TRUMAN, The University of New Mexico — We present experimental results obtained by shock acceleration of a gravity- and co-flow-stabilized cylindrical heavy gas (SF_6) jet embedded in lighter gas (air). The angle between the plane of the shock and the axis of the cylinder was varied between zero (planar interaction) and fifteen degrees (oblique interaction). The Mach number M varied from 1.2 to 2.4, with most of the data acquired at $M = 2$. We simultaneously acquired two views of the resulting flow – top and side, using diffuse white light to visualize Mie scattering in submicron-sized droplets carried by the cylindrical jet. Our observations show that in the case of the planar shock-jet interaction, the boundary layers on the walls of the shock tube where the experiment was conducted do not appear to play a dramatic role. In the case of the oblique interaction, however, vorticity deposition produced by the shock interaction with the density gradients apparently leads to the jet material visibly interacting with one of the wall boundary layers.

[1]This research is supported by the United States Defense Threat Reduction Agency (DTRA), award numbers HDTRA1-07-1-0036 and HDTRA1-08-1-0053.

11:22AM HP.00005 Shock-resolved Navier-Stokes Simulation of the Richtmyer-Meshkov Instability , RICHARD KRAMER, University of Illinois at Urbana-Champaign, DALE PULLIN, California Institute of Technology — Results are presented from a numerical investigation of the Richtmyer-Meshkov instability, using a first-order perturbation of the two-dimensional Navier-Stokes equations about a one-dimensional unsteady shock-resolved base flow. This approach captures perturbations on the shocks and their influence on the interface growth to accurately examine the start-up and early linear growth phases of the instability. Weak, intermediate and strong incident shocks cases are examined for a single fluid and an Air/SF_6 mixture, across a range of contact-zone perturbation wave numbers, and compared to analytic models for transient and asymptotic growth of the instability.

11:35AM HP.00006 On the Simulation of Shock-Driven Material Mixing in High Reynolds-Number Flows[1] , FERNANDO GRINSTEIN, AKSHAY GOWARDHAN, Los Alamos National Laboratory — Implicit large eddy simulation proposes to effectively rely on the use of subgrid modeling and filtering provided implicitly by physics capturing numerics. Extensive work has demonstrated that predictive simulations of turbulent velocity fields are possible using a class of high resolution, non-oscillatory finite-volume (NFV) numerical algorithms. Truncation terms associated with NFV methods implicitly provide subgrid models capable of emulating the physical dynamics of the unresolved turbulent velocity fluctuations by themselves. The extension of the approach to the substantially more difficult problem of under-resolved material mixing by an under-resolved velocity field has not yet been investigated numerically, nor are there any theories as to when the methodology may be expected to be successful. Progress in addressing these issues in studies of shock-driven scalar mixing driven by Ritchmyer-Meshkov instabilities will be reported in the context of ongoing simulations of shock-tube laboratory experiments.

[1]Supported by LANL LDRD-DR on "Turbulence by Design."

11:48AM HP.00007 Simulation of Material Mixing in Shocked Gas-Curtain Experiments[1] , AKSHAY GOWARDHAN, FERNANDO GRINSTEIN, Los Alamos National Laboratory — 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. An SF6 gas curtain is formed by forcing SF6 through a linear arrangement of round nozzles into the shocktube test section. The gas curtain is shocked (M=1.26, M=1.5), and its later evolution subject to Ritchmyer-Meshkov flow instabilities, transition, and non-equilibrium turbulence phenomena are investigated based on high resolution simulations for shocked and reshocked cases. The particular strategy tested here is based on a nominally-inviscid simulation model using the LANL RAGE code and adaptive mesh refinement. Initial conditions for ILES are based on emulating the physics of SF6 falling through the test section of the shock tube until a steady state is reached using a separate 3D Navier-Stokes code which solves incompressible flow in the Boussinesq approximation.

[1]Supported by LANL LDRD-DR on "Turbulence by Design"

12:01PM HP.00008 Mixing asymmetry in variable density turbulence , DANIEL LIVESCU, RAY RISTORCELLI, ROBERT GORE, Los Alamos National Laboratory — The homogenization of a heterogeneous mixture of two pure fluids with different densities by molecular diffusion and stirring induced by buoyancy generated motions is studied using Direct Numerical Simulations in two configurations: a) classical Rayleigh-Taylor instability using a 3072^3 data set (Cabot and Cook, Nature Phys. 2006, Livescu et al, J. Turb. 2009) and b) an idealized triply periodic Rayleigh-Taylor flow named hereafter homogeneous Rayleigh-Taylor (HRT), using up to 1024^3 meshes (Livescu and Ristorcelli, J. Fluid Mech. 2007, 2008). As a consequence of the differential accelerations experienced by the fluids, important differences between the mixing in a variable density flow, as compared to the Boussinesq approximation, are observed. In short, the pure heavy fluid mixes more slowly than the pure light fluid: in HRT, an initially symmetric double delta density PDF is rapidly skewed, as the light pure fluid vanishes, and only at long times and small density differences it relaxes to a symmetric, Gaussian-like PDF. The effect is shown to be related to the local structure of the flow and consequences for the high Atwood number Rayleigh-Taylor mixing are discussed.

12:14PM HP.00009 Initial conditions effects in shock-driven instabilities , SRIDHAR BALASUBRAMANIAN, B.J. BALAKUMAR, GREG ORLICZ, CHRIS TOMKINS, KATHY PRESTRIDGE, LANL — Recent work at LANL (Dimonte et al, Phys Fluids 2004, Ramaprabhu et al, JFM 2005) has shown that buoyancy-driven turbulence can be affected at late-time by initial conditions, and memory of the initial conditions is not lost. We study the initial condition parameters that impact the mixing and transition to turbulence in shock-driven, Richtmyer-Meshkov unstable flows. A detailed study of the impact of wavelength and amplitude of initial condition perturbations in a heavy gas curtain (air-SF6-air) is undertaken. Carefully controlled, membrane-free initial conditions with At=0.67 and shock Mach number of 1.2 are used to examine the effect of varying the initial modes of the gas curtain. The temporal and spatial evolution of the flow is measured using simultaneous Particle Image Velocimetry (PIV) and Planar-Laser Induced Fluorescence (PLIF). This work is supported by the Los Alamos Laboratory Directed Research and Development Program.

12:27PM HP.00010 Numerical Study of Multi-Component Mixing in Shock-Accelerated Flows Using Localized Artificial Diffusivity Method[1] , SANTHOSH SHANKAR, SOSHI KAWAI, SANJIVA LELE, Stanford University — There has been a long-standing interest in developing numerical algorithms for compressible multi-component flows. The performance of the localized artificial diffusivity method of Kawai et al. (JCP 2009) on such problems is demonstrated by comparing the results for simple test cases with exact solutions and previous numerical calculations. The scheme is then used to simulate a 2D shock-bubble interaction by solving the full compressible Navier-Stokes equations to study the underlying mechanisms of mixing in this flow. The initial concentration of the dense gas bubble is modeled and the initial flow conditions are matched to the experiments of Tomkins et al. (JFM 2008) where a Mach 1.2 shock in air interacts with a cylindrical column in SF6. The grid converged results of the mixing rate and profile are quantitatively compared with the experiments for validation. In addition 3D simulations of Richtmyer-Meshkov instability based on the experiments of Vetter and Sturtevant (Shock Waves 1995) will be presented with particular focus on the turbulent missing in terms of the mixing width, the mixing rate and turbulent spectra.

[1]Supported by DOE-SCIDAC and NASA Hypersonics project.

Monday, November 23, 2009 10:30AM - 12:27PM –
Session HQ Instability: Interfacial and Thin-Film V 200E

10:30AM HQ.00001 Drop formation from an unstable partially wetting fluid rivulet , JAVIER A. DIEZ, ALEJANDRO G. GONZÁLEZ, Instituto de Física Arroyo Seco, Universidad Nacional del Centro de la Provincia de Buenos Aires, Pinto 399, 7000, Tandil, Argentina, LOU KONDIC, Department of Mathematics, New Jersey Institute of Technology, Newark, NJ, PHILIP D. RACK, Department of Materials Science and Engineering University of Tennessee, Knoxville, TN — We consider the formation of drops by the spontaneous breakup of an unstable liquid rivulet on a horizontal substrate under partial wetting conditions. We describe the dynamics by means of a model within the lubrication approximation that includes capillarity, van der Waals forces, and gravity. We focus on gravity effects for macroscopic rivulets, and on intermolecular forces for the nanoscopic case. We find the scaling law of the emerging distance between drops (formed after the breakup process) as a function of the rivulet cross-section area. Unlike the case of thin films, the numerical results for finite length rivulets show that there is no nucleation regime, and only spinodal instability develops. Finally, we apply this model to the study of a metalic rivulet of nanometric thickness, melted via pulsed laser radiation.

10:43AM HQ.00002 Droplet deformation in a channel: viscoelastic effects[1] , ARTHUR PERCHERON, ESPCI, Paris, ROUSHAN ALAM, GITA SEEVARATNAM, JERRY HENG, OMAR MATAR, Imperial College London — In this work, the influence of viscoelasticity on the deformation of a sessile droplet subjected to a pressure-driven flow in a channel is investigated experimentally. The droplet's response is examined using flow visualisation as a function of initial droplet volume, flow rate and elasticity. The results of our experiments are used to identify various flow regimes. These include low flow rate regimes wherein the droplets optimise their shapes in response to the flow before reaching a steady state. With increasing flow rate, droplets exhibit "sliding", "crawling", and detachment regimes. In the latter case, and due to large elastic contributions, droplets develop very long necks, which pinch off and recoil to join the remnants of the mother droplet. By plotting a "flow map" in the space of the Ohnesorge and Deborah numbers, transitions between each regime of deformation are identified.

[1]EPSRC Grant number EP/E046029/1

10:56AM HQ.00003 Surfactant-enhanced rapid spreading of drops on solid surfaces[1] , DAVID BEACHAM, Imperial College London, RICHARD CRASTER, University of Alberta, OMAR MATAR, Imperial College London — We consider the surfactant-enhanced rapid spreading of drops on solid substrates. This work is conducted in connection with the ability of aqueous trisiloxane solutions to wet effectively highly hydrophobic substrates. We use lubrication theory to derive coupled advective-diffusion equations for surfactant transport to an interface equation. This model accounts for Marangoni stresses, diffusion, intermolecular forces, basal surfactant transport, micelle formation and break-up in the bulk, and sorptive fluxes at both the gas-liquid and liquid- solid interfaces; the model also employs appropriate surfactant equations of state. Our numerical results show the effect of basal adsorption and the mass of deposited surfactant on the deformation of the droplet and its spreading rate. We demonstrate that this rate is maximised for intermediate rates of basal adsorption and total surfactant mass. We also show that for a certain range of parameter values, the spreading is accompanied by pronounced rim formation, as previously observed experimentally. The stability of this rim to transverse disturbances is briefly explored.

[1]EPSRC for DTA studentship for DRB

11:09AM HQ.00004 Dynamics of surfactants spreading on gel layers[1] , CONSTANTINE SPANDAGOS, PAUL LUCKHAM, OMAR MATAR, Imperial College London — Gel-like materials are of central importance to a large number of engineering, biological, biomedical and day-life applications. This work attempts to investigate the spreading of droplets of surfactant solutions on agar gels, which is accompanied by cracking of the gel layers. The cracking progresses via the formation of patterns that resemble "starbursts," which have been reported recently in the literature by Daniels et al. Marangoni stresses generated by surface tension gradients between the surfactant droplet and the uncontaminated gel layer are identified to be the driving force behind these phenomena. The morphology and dynamics of the starburst patterns are investigated for droplets of different surfactant solutions, including sodiumdodecylsulphate, spreading on gel layers of different strengths. The instability is characterised in terms of the number of arms that form, and their mean width and length as a function of time. In addition, photoelasticity is used to provide information about the stress field of the material, which, combined with the results from our direct visualisation, can elucidate further the mechanisms underlying the pattern formation and the nature of the interactions between the liquid and the gel.

[1]EPSRC Grant number EP/E056466

11:22AM HQ.00005 Effects of dilution on elastohydrodynamic coating flow of an anti-HIV microbicide vehicle[1] , ANDREW SZERI, SU CHAN PARK, SAVAS TASOGLU, Department of Mechanical Engineering, UC Berkeley, DAVID F. KATZ, Department of Biomedical Engineering and Department of Obstetrics and Gynecology, Duke University — Elastohydrodynamic lubrication over soft substrates characterizes the drug delivery of anti-HIV topical microbicides carried in gel vehicles. These gels are under development to prevent HIV transmission into vulnerable vaginal mucosa during intercourse. Their effectiveness depends on completeness and durability of coating, as well as on the active ingredients. Here we investigate the influence of dilution by vaginal fluid on the coating flows that serve to protect the user. The effects of dilution by vaginal fluid simulant are assessed through rheological experiments at variable dilution of the gel vehicle. This involves determination of the way parameters in a Carreau model of a shear-thinning gel are modified by dilution. The changes in coating are determined from a computational model, based on dilution rheology measured in the laboratory. The elastohydrodynamic lubrication model of Szeri, et al. Physics of Fluids (2008) is supplemented with a convective-diffusive transport equation to handle dilution, and solved using a multi-step scheme in a moving domain.

[1]Work supported by the California HIV/AIDS Research Program.

11:35AM HQ.00006 ABSTRACT WITHDRAWN –

11:48AM HQ.00007 Lift-Force-Driven Microfluidic Droplet Sorting Device , HARM NIEUWSTADT, DAVID LI, ROBINSON SEDA, J. BRIAN FOWLKES, JOSEPH BULL, The University of Michigan — A long ($\sim$ 5-7 centimeters) rectangular micro channel is used to sort perfluorcarbon (PFC) droplets by size.This study is motivated by a novel gas embolotherapy technique which aims to treat cancer by infarcting tumors with gas emboli that are formed by selective acoustic vaporization of $\sim$6 micrometer, intravascular, PFC droplets. Droplets smaller and larger than $\sim$6-micrometer proved to be less effective, or even detrimental, in the gas embolotherapy. From a suspension of micro PFC droplets, the sorting device designed in this study eliminates droplets with higher and lower diameters than $\sim$6-micrometer. This is done by the use of a difference in lateral lift force (which depends on droplet diameter) that is exerted on the droplets. When a mixture of droplets is introduced at the entrance of a straight micro channel, larger droplets will tend to move to an equilibrium position at 0.6R, R being the half width of the channel, in shorter time and distance than smaller droplets. When splitting up the end of the channel in two smaller outer channels (which contain the large droplets) and one smaller inner channel (which contains the small droplets), the droplets can, in theory, be sorted with high accuracy. This work is supported by NIH grant R01EB006476.

12:01PM HQ.00008 Viscous Fingering of Reactive Finite Slices in a Hele-Shaw Cell[1] , S. HOSSEIN HEJAZI, JALEL AZAIEZ, University of Calgary — Coupling of hydrodynamics and chemical reactions have been studied for a miscible reactive slice of fluid displaced by another fluid in a rectilinear Hele-Shaw cell. Under this configuration, one of the two reactants is sandwiched between two layers of the other one. Assuming an exponential dependency of viscosity on the solution concentrations, the fluid fronts at both ends of the finite slice may become unstable. Full nonlinear simulations are conducted to solve the continuity equation, the momentum balance equation in the form of Darcy's law, and a volume-averaged mass balance equation in the form of convection-diffusion-reaction equations for the transport of solute and solutions. Simulation results indicate the importance of the frontal instabilities on the disappearing rate of the reactant fluid in the slice. The effects of the viscosity ratios between the two reactants and chemical product on the finger structures, the slice distortion and chemical reaction progress are examined both qualitatively and quantitatively.

[1]The authors acknowledge the financial support by Alberta Ingenuity Fund (AIF), the Alberta Ingenuity Centre for In-Situ Energy (AICISE) and also the use of the computing resources of the West-Grid cluster.

12:14PM HQ.00009 Influence of miscible viscous fingering on an adsorbed solute dynamics , MANORANJAN MISHRA, Universie Libre de Bruxelles, Brussels, Belgium, MICHEL MARTIN, PMMH-ESPCI, Paris, France, ANNE DE WIT, Universie Libre de Bruxelles, Brussels, Belgium — Viscous fingering between miscible fluids of different viscosities can affect the dispersion of finite width samples in porous media. We investigate here the influence of such VF due to a difference between the viscosity of the displacing fluid and that of the sample solvent on the spatio-temporal dynamics of the concentration of a passive solute initially dissolved in the injected sample and undergoing adsorption on the porous matrix. Such a three component system is modeled using Darcy's law for the fluid velocity coupled to mass-balance equations for the sample solvent and solute concentrations. Depending on the conditions of adsorption, the spatial distribution of the solute concentration can either be deformed by viscous fingering of the sample solvent concentration profiles or disentangle from the fingering zone. In the case of deformation by fingering, a parametric study is performed to analyze the influence of parameters such as the log-mobility ratio, the ratio of dispersion coefficients, the sample length and the adsorption retention parameter k' on the widening of the solute concentration peak.

Monday, November 23, 2009 10:30AM - 12:40PM –

Session HR Porous Media Flows 200F

10:30AM HR.00001 The fate of the sessile droplet imprint in porous medium: simultaneous capillary flow and evaporation , B.I. MARKICEVIC, H.K. NAVAZ, Kettering University — The fate of a liquid droplet imbibed into a porous medium is formulated as a multiphase problem, and a numerical solution is developed using the capillary network model with a micro-force balance at the liquid|gas interface. Momentum transport - capillary flow, and mass transport - evaporation are solved simultaneously. The physics of the multiphase capillary flow includes the formation of local gas clusters, and liquid ganglia, whose distribution can be determined from the force balance on the gas|liquid interface. The clusters and ganglia distribution is further altered by evaporation. The evaporation tends to shrink the ganglia size and open the gas clusters; both due to the liquid mass loss from the porous medium. Still, the capillarity tends to disperse the liquid back into the regions from where the liquid previously evaporated. In order to quantify the liquid distribution, besides the diffusion coefficient, the dispersion coefficient for the capillary flow is defined. The latter is found from the porous medium permeability, liquid viscosity and capillary pressure. As expected, for a larger dispersion coefficient, the liquid remains closer to the evaporating boundary, and the evaporation rate is higher. The opposite is true for a small dispersion coefficient. Finally, the changes in liquid dispersion influence the liquid persistence time, where this time increases for a liquid dispersed deeper in the medium.

10:43AM HR.00002 Capillary Flow Limitations of Nanowicks , CONAN ZHANG, CARLOS HIDROVO, University of Texas-Austin — Thermal management is an important issue in microelectronic systems. The inaccessibility and diminishing size of these systems, however, requires that the heat management components be reliable and compact, such as is the case with heat pipes. In most intermediate temperature heat pipes typically found in microelectronics, the critical heat flux is governed by the capillary limit. Given the projected increases in computer chip heat fluxes, it is important to investigate the use of nanowicks as a means of raising this capillary limit. A theoretical model was developed to simulate flow through a vertical nanopillar array by balancing the capillary driving forces and the viscous losses in a quasi-steady state dynamic formulation. Based on this model, the maximum mass flow and its critical heat flux can be found for a wick given its microstructure geometry. These values were also found experimentally for commercially available wicks and nanowicks. We found that nanowicks provide lower mass flow rates than conventional wicks, mainly due to a reduced cross section. However, nanowicks achieved higher velocities and show promise over some conventional heat pipe wicks.

10:56AM HR.00003 Thermo-chemical energy storage and heat transfer in a flow of hydrated magnesium sulfate , GANESH BALASUBRAMANIAN, Department of Engineering Science and Mechanics, Virginia Polytechnic Institute and State University, SOHAIL MURAD, Department of Chemical Engineering, University of Illinois at Chicago, ISHWAR K. PURI, Department of Engineering Science and Mechanics, Virginia Polytechnic Institute and State University — Salt hydrates undergo desorption on being heated above certain charging temperatures, releasing water and forming anhydrous salts which have a higher energy content. Since these salts are hygroscopic, energy is easily retrieved back by passing water vapor over the anhydrous form. Such a technique of energy conversion, storage and retrieval enables these salts to be impregnated into porous media for thermo-chemical energy application. However, to investigate the thermal transport at the interface of the porous material and the salt, atomistic simulations are necessary. We employ molecular dynamics to simulate the heat transfer mechanism in a flow of hydrated magnesium sulfate impregnated into mesoporous silica and understand the role of interfacial thermal resistance on the charging temperature and total heat storage capacity of such salts.

11:09AM HR.00004 Drainage in Gas diffusion Layers of PEM fuel cells , EZEQUIEL MEDICI, JEFF ALLEN, Michigan Technological University — Percolation through porous transport layers (PTL) of proton exchange membrane (PEM) fuel cells have distinct fluid flow patterns depending on the PTL morphological and wetting properties as well as injected/displaced fluid properties and flow rates. These distinct fluid flow patterns include stable displacement, capillary fingering, and viscous fingering. A pseudo Hele-Shaw cell experimental setup was developed to effectively characterize different gas diffusion layers based on their susceptibility to the formation of these fluid flow patterns. The wetted area is determined by tracking the injected fluid from a top view of the cell. The percolation pressure is measured at the injection location of the cell. These two outputs are essential to identify the mechanism controlling the percolation process. The goal of this work is to link the wetted area and percolation pressure through non-dimensionalization that collapses these onto a single curve. Preliminary results from capillary fingering and stable displacement data show a constant shift between the curves for different PTL samples. This allows for unique characterization of PTLs based on their morphological and wettability properties.

11:22AM HR.00005 ABSTRACT WITHDRAWN —

11:35AM HR.00006 Natural convection and the evolution of a reactive porous medium[1] , LINDSEY RITCHIE, DAVID PRITCHARD, University of Strathclyde — In many geological settings the equilibrium concentration of a dissolved chemical species depends on the vertically varying chemical properties of the rock. As a prototype for such systems, this study considers the solutal convection of a reactive fluid in a permeable medium, subject to chemical equilibrium on the bounding surfaces, and where the chemical equilibrium varies vertically throughout the layer. Over relatively short timescales, the exchange of solute between fluid and matrix stabilises the system against the onset of convection and promotes the development of narrower convective cells: we investigate this process using linear stability analysis and numerical simulation. Over longer timescales, when convection has developed, the reaction drives spatial and temporal changes to the porosity and permeability of the rock. We describe numerical results which reveal novel interactions between the convection pattern and the evolving matrix. Ultimately these interactions can restabilise the system, shutting down the convection but preserving "signatures" of convective cells or more complex structures in the varying porosity of the rock.

[1]The Carnegie Trust for the Universities of Scotland provided support for this work.

11:48AM HR.00007 Post-Injection Migration of CO_2 in Saline Aquifers subject to Groundwater Flow, Aquifer Slope, and Capillary Trapping , CHRISTOPHER MACMINN, MICHAEL SZULCZEWSKI, RUBEN JUANES, Massachusetts Institute of Technology — We study a sharp-interface mathematical model for the post-injection migration of a plume of CO_2 in a deep saline aquifer under the influence of natural groundwater flow, aquifer slope, gravity override, and capillary trapping. The model leads to a nonlinear advection-diffusion equation, where the diffusive term describes the upward spreading of the CO_2 against the caprock. We find that the advective terms dominate the flow dynamics even for moderate gravity override, and we solve the model analytically in the hyperbolic limit. We use this solution to estimate the capacity of various target formations in the United States.

12:01PM HR.00008 Uncertainty characterization of solubility trapping of CO2 in saline aquifers , AMIR RIAZ, University of Maryland, MARC HESSE, University of Texax Austin, HAMDI TCHELEPI, Stanford University — Buoyancy driven unstable convection of CO2 saturated plumes in brine aquifers that results from the Rayleigh-Benard hydrodynamic instability in a porous medium is one of the principle mechanisms of CO2 sequestration by solubility trapping. The finite time of the onset of instability has been analyzed in detail by many authors previously but an agreement as to its precise value as a function of the Rayleigh number has not yet emerged. Rather, the onset time obtained from various analyses is known to vary by an order of magnitude. In an attempt to resolve this discrepancy we carry out high accuracy numerical simulations and measure the critical time. We find that the critical time and the closely related time-to-initiate-convection are fundamentally dependent on the magnitude and realization of the field of initial random perturbations employed to induce instability and are hence not deterministic. We therefore provide a probabilistic description of the processes of initialization of instability and convection and formulate correlations with late time events such as the total amount of CO2 dissolved to provide a reliable characterization of the process of CO2 solubility trapping in brine aquifers.

12:14PM HR.00009 Gravity currents propagating along channels in porous media[1] , MADELEINE GOLDING, HERBERT HUPPERT, University of Cambridge, INSTITUTE OF THEORETICAL GEOPHYSICS, DEPARTMENT OF APPLIED MATHEMATICS AND THEORETICAL PHYSICS TEAM — The effect of impermeable channel boundaries on gravity currents propagating in a porous medium is investigated. The problem admits similarity solutions for currents of a variety of rates of input flux and channel shapes. Experiments were conducted in V-shaped and semicircular channels and the results are generally found to be in good agreement with theoretical predictions. One motivation for the study of gravity currents in a porous medium is the process of carbon dioxide (CO_2) sequestration, whereby supercritical CO_2 is pumped deep underground into rock saturated with denser salt water. The CO_2 rises as a buoyant plume until it reaches an impermeable boundary, at which point it spreads laterally as a gravity current. Our knowledge of geological structure deep underground is limited. Therefore the theory developed in this study aims to help understand to what extent the presence of more complex boundaries, namely channel boundaries, would affect predictions for the propagation of gravity currents in porous media.

[1]The research of M.J.G. is funded by the EPSRC and that of H.E.H. is partially supported by a Royal Society Wolfson Research Merit Award.

12:27PM HR.00010 Localised leakage from porous and viscous gravity currents , JEROME NEUFELD, DOMINIC VELLA, HERBERT HUPPERT, JOHN LISTER, ITG, DAMTP, University of Cambridge — Motivated by the geological sequestration of CO_2 we investigate the effect of localized leakage on the spreading of porous and viscous gravity currents in a variety of both two and three dimensional geometries. Generically, we find that these systems tend towards a steady state in which the input flux is exactly balanced by the leakage flux. Of particular importance is the efficiency of storage defined as the instantaneous fraction of fluid that does not leak. We compare analytic expressions for the asymptotic behavior of the efficiency to full numerical results and laboratory models of both porous and viscous gravity currents. These results bear on the time scales over which CO_2 may be safely stored in saline aquifers.

Monday, November 23, 2009 10:30AM - 12:27PM –
Session HS Geophysical: Oceanographic III 200G

10:30AM HS.00001 Numerical Simulations of Breaking Waves , DOUGLAS DOMMERMUTH, THOMAS O'SHEA, KYLE BRUCKER, SAIC — The results of a set of numerical simulations of breaking waves are presented. The breaking waves are generated using a novel type of atmospheric forcing, which allows for wave breaking events characterized by strong plunging to weak spilling. The analysis of the wave breaking events consists of single point quantities, two-dimensional statistical quantities obtained by averaging, and three dimensional isocontours of instantaneous field variables. The single point quantities investigated include the kinetic and potential energies integrated over the computational volume. The effect of the atmospheric forcing on the late time asymptotic behavior of the kinetic and potential energies will be discussed. In terms of the two-dimensional statistical quantities: The use of binary and ternary averaged statistical quantities are compared, and it is found that statistics obtained by ternary averaging show better agreement with respect to the position of free surface than do those obtained by binary averaging. The amount of air entrained underneath the free surface is investigated both qualitatively and quantitatively. The three dimensional isocontours of vorticity show intense streamwise vorticity, or tubular structures, apparent during the strong plunging events.

10:43AM HS.00002 Wave breaking onset and spectral distribution of energy loss due to laboratory generated breaking waves , ZHIGANG TIAN, MARC PERLIN, Naval Architecture and Marine Engineering, University of Michigan, Ann Arbor, MI 48109 USA, WOOYOUNG CHOI, Department of Mathematical Sciences, New Jersey Institute of Technology, Newark, NJ 07102 USA — A study of wave breaking onset and energy loss due to wave breaking is presented. A global wave steepness parameter that accounts for spectral amplitude shape change is proposed. This parameter has a threshold of 0.25 for wave breaking onset and it correlated strongly with breaking energy loss. It exhibits reduced dependence on the shape of the wave spectrum, and it may be a better indicator for breaking onset and energy loss. Spatial evolution of frequency spectra is presented. Noticeable energy loss due to viscous effects is observed near the spectral peak. Spectral redistribution of energy due to wave breaking is presented. High frequency waves lose energy while low frequency components gain as much as 50% of that energy loss. Energy near the spectral peak may increase or decrease, and is a function of the initial spectrum. In addition the change in the spatial distribution (i.e. wavenumber spectra) of energy loss will be presented.

10:56AM HS.00003 A propagation model for internal waves generated by a localized source moving in an ocean thermocline , JAMES ROTTMAN[1], Science Applications International Corporation, DAVE BROUTMAN, Computational Physics, Inc. — We develop a ray method for computing the propagation of internal waves generated by a localized source moving in an ocean thermocline. Previous work on this problem has involved an eigenfunction expansion. Eigenfunctions are often preferred over rays for regions such as the thermocline that have many ray reflections. Here we show that a ray method has some computational advantages over eigenfunctions, provided the rays are traced in Fourier space. The ray method does not require an eigenfunction solver and does not have the singularites that appear in the eigenfunction expansion. It does, however, require the correction of caustic singularities. For the thermocline problem, each ray has two caustics, one at each of the two the turning points that vertically trap the ray. Results for various buoyancy-frequency profiles will be presented and compared with previous computational techniques.

[1]Funded by ONR contract # N00014-08-C-0508.

11:09AM HS.00004 Numerical simulation of shear instabilities in interfacial gravity waves[1] , OLIVER FRINGER, MICHAEL BARAD, Stanford University — We present simulations of shear instabilities in solitary-like interfacial gravity waves of depression using a Navier-Stokes solver that employs adaptive mesh refinement. The adaptive technique enables resolution of 0.20 m in a 500 m long wave which allows simulation of meter-scale Kelvin-Helmholtz (KH)-like billows that develop at the interface. In the presence of time-varying shear within the waves, an instability occurs only when a parcel of fluid is subjected to destabilizing shear long enough for KH-type billows to grow. While a necessary criterion for instability suggests that the Richardson number, Ri, must fall below the canonical value of $1/4$, we find that a sufficient condition for instability occurs when $Ri < 0.1$. An alternate criterion for instability is given by a requirement that the growth rate time scale of the instability, τ_i, satisfies $\tau_i < 1.26T_w$, where T_w is the time in which parcels of fluid are subjected to shear and stratification that satisfy $Ri < 1/4$. This criterion can also be stated in terms of the width of the region in which $Ri < 1/4$, L_w, which must satisfy $L_w > 0.86L$ for instabilities to develop, where L is the solitary wave half-width. Under any one of these three criteria, two-dimensional billows form at the wave troughs, and the billows subsequently break down via three-dimensional motions that decay once the wave-induced shear subsides in the trailing edge of the waves.

[1]Supported by the ONR and an NSF postdoctoral fellowship.

11:22AM HS.00005 Near-bottom instabilities induced by nonlinear internal waves of depression[1] , THEMISTOKLIS STEFANAKIS, PETER DIAMESSIS, Cornell University — Results are presented from 2-D Direct Numerical Simulations (DNS) of the bottom boundary layer (bbl) in the footprint of fully nonlinear internal waves (NLIWs) of depression propagating in a uniform-depth two-layer system. Use of a spectral multidomain penalty method enables a robust and accurate description of the wave-induced bbl at values of Reynolds number (based on NLIW phase speed and wave-guide depth) as high as 100,000. The critical NLIW amplitude for global instability in the NLIW-driven bbl is identified as a function of Reynolds number, oncoming current strength and layer thickness ratio. The structure of of bbl vorticity and bottom shear stress fields is examined along with timeseries obtained of the near-bed vertical velocity fields obtained from Lagrangian and Eulerian virtual point sensors. DNS results are compared to recent laboratory observations, followed by a discussion of the underlying implications for the field.

[1]Funded by the Office of Naval Research.

11:35AM HS.00006 Backscatter in Large-Scale Flows , BALU NADIGA, LANL — Downgradient mixing of potential-voriticity and its variants are commonly employed to model the effects of unresolved geostrophic turbulence on resolved scales. This is motivated by the (inviscid and unforced) particle-wise conservation of potential-vorticity and the mean forward or down-scale cascade of potential enstrophy in geostrophic turubulence. By examining the statistical distribution of the transfer of potential enstrophy from mean or filtered motions to eddy or sub-filter motions, we find that the mean forward cascade results from the forward-scatter being only slightly greater than the backscatter. Downgradient mixing ideas, do not recognize such equitable mean-eddy or large scale-small scale interactions and consequently model only the mean effect of forward cascade; the importance of capturing the effects of backscatter—the forcing of resolved scales by unresolved scales—are only beginning to be recognized. While recent attempts to model the effects of backscatter on resolved scales have taken a stochastic approach, our analysis suggests that these effects are amenable to being modeled deterministically.

11:48AM HS.00007 Extension of Brinkman Penalization for Ocean Circulation Modeling using Adaptive Wavelet Collocation Method[1] , SHANON RECKINGER, OLEG V. VASILYEV, University of Colorado — This work improves the representation of continental topology and bottom bathymetry for use in ocean circulation models 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 large accuracy and stability problems. Brinkman penalization is a numerical technique used to enforce no slip boundary conditions through the addition of a term to the governing equations. When coupled with the Adaptive Wavelet Collocation Method, which solves the penalized equations on a temporally and spatially varying mesh, the flow near the boundary can be well defined for less computational cost. The talk presents an extension of this technique to slip boundary conditions and is applied to the various sets of equations that govern ocean circulation. The solution using this new penalization converges to the numerical solution with traditional boundary conditions similarly to the order η convergence of the Brinkman penalization method that enforces no slip conditions.

[1]This work was supported by DOE-CCPP (DE-FG02-07ER64468).

12:01PM HS.00008 Detached eddy simulation of separated flow in the near-wake of a sphere moving through a stratified fluid[1] , T.S. ORR, J.A. DOMARADZKI, University of Southern California, G.S. CONSTANTINESCU, University of Iowa — Separated turbulent flows generated by submersed bodies in stratified fluids have been investigated in numerous experiments and numerical simulations. At high Reynolds numbers the numerical simulations are unable to resolve both a flow around the body and in the far-wake, which is the region of interest in such investigations. Because of that, in the far-wake simulations, the body is not simulated directly, but an initial condition is constructed using limited information from experiments for the near-wake properties. However, such initializations suffer from the lack of information about density field, which is difficult to measure. To obtain such information we perform DES of a flow around a sphere moving through a stably stratified fluid employing a body-fitted viscous flow solver in generalized curvilinear coordinates. The DES method based on the Spalart-Allmaras turbulence model is used to model the momentum equation as well as the density perturbation equation in the Boussinesq approximation. The simulations are validated by comparison with experimental data collected at USC at a $Re = 5000$ and $Fr = 4$ and then parameter space is extended to cover several other values of Re and Fr of interest.

[1]Supported by ONR

12:14PM HS.00009 New Lagrangian tools for describing transport in aperiodic time dependent flows: a case study of the Kuroshio current[1] , ANA M. MANCHO, CAROLINA MENDOZA, ICMAT, CSIC — In recent years there has been a lot of progress in the application of dynamical systems concepts to the description of transport in oceanic flows. In these flows the classical dynamical system theory does not apply since they are aperiodic and finite-time defined. Recently, for describing these flows a new definition of distinguished trajectory has been proposed (Madrid & Mancho, Chaos, 2009). Distinguished trajectories act as organizing centres of the geometrical template of aperiodic time-dependent flows, like fixed points and periodic orbits do in time independent or periodic flows. The computation of distinguished trajectories makes use of a function M of which we show contains a lot of Lagrangian information. In this presentation I will discuss how the visualization of this function M, allows identifying relevant Lagrangian features at a glance. In particular we report an application to real altimetry data taken from satellite in the area of the Kuroshio current. The function M also determines the stable and unstable subspaces of the distinguished hyperbolic trajectories which are tangent to the invariant manifolds. From the computation of stable and unstable manifolds we report an accurate description of transport routes in this region.

[1]We acknowledge computational support by CESGA.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HT Vortex Dynamics and Vortex Flows V 200H

10:30AM HT.00001 Global Description of Bifurcation Branches and Nonlinear Dynamics of Vortex Flow in a Pipe[1] , ANSHUMAN MISHRA, GILEAD TADMOR, Northeastern University, BERND R. NOACK, Berlin Institute of Technology, ZVI RUSAK, Rensselaer Polytechnic Institute — Details of the global dynamics of the transition to vortex breakdown in the high Reynolds number, high swirl, axisymmetric vortex flow in a finite pipe, are investigated. A first global map of fixed point branches that bifurcate from the columnar flow is revealed, along with a detailed characterization of the nonlinear, local dynamics near fixed points. The description includes the fixed point states, linear and nonlinear stability analysis modes that dominate the fluctuations near fixed points, secondary bifurcations along branches, and dynamic interconnections between them. A central role in the investigation is played by low and least order, mean-field Galerkin models of the local dynamics near bifurcation branches. Indeed, as this is a first case study of mean field Galerkin models in a flow configuration with multiple coexisting attractors, the technical and conceptual aspects of these models and of their identification, are of independent interest. The unveiled picture is that of an inertial manifold that is well approximated with only few dominant coherent structures, or modes that are well characterized by temporal and spatial frequencies, and that continuously deform with changes in the operating conditions.

[1]The research was supported by NSF grants 0524070 and 0523957

10:43AM HT.00002 Vortex breakdown of a swirling light gas jet discharging into a heavier ambient gas[1] , J.M. GALLARDO, C. DEL PINO, R. FERNANDEZ-FERIA, U. Malaga (Spain) — The effect of co-flow in the structure of laminar gas swirling jets with very small jet-to-ambient density ratios has been investigated by solving the high Reynolds number parabolic equations. The study is of interest for the design of Hydrogen swirl combustors. We find that the critical swirl number for vortex breakdown decreases for increasing co-flow ratios, as in a swirling jet discharging into the same ambient gas, but the critical swirl for breakdown in a light gas jet can be significantly larger. As the co-flow increases, the difference between both critical swirl numbers decreases, are equal for co-flow ratio unity, and, for co-flow ratios larger than unity, the critical swirl for a light gas jet becomes smaller than that for a homogeneous jet. These behaviors are explained by the differences in the pressure distributions generated by the swirl when the jet is much lighter than the ambient gas, in relation to a homogeneous jet. The situation becomes more complex when compressibility effects are taken into account, owing to the interplay between temperature, density and pressure differences generated by the swirl. We characterize the critical swirl for vortex breakdown as a function of the density ratio, the co-flow ratio, and the Mach number.

[1]Supported by the Min. Ciencia Innov. (Spain) Grant no. FIS07-60161.

10:56AM HT.00003 Interaction of multiple solid objects and vortices in a two-dimensional potential flow , J.T. WU, Department of Civil Engineering, National Taiwan University, F.L. YANG, D.L. YOUNG — The generalized Lagally theorem states that the unsteady hydrodynamic force acting on a rigid body exhibiting arbitrary motion in an inviscid and incompressible fluid can be expressed explicitly in terms of singularities such as sources and dipoles which generate the flow field. This work extends the theorem by considering the presence of point vortices in the flow, which results in additional force components when formulating the equation of motion for the solid bodies. The present model has been applied to describe the motion of a circular cylinder that interacts with a vortex. The obtained result is in exact agreement with the result derived by other researchers considering the conservation of the sum of the solid momentum and the fluid impulse. The current model is further applied to study a moving cylinder towards a fixed one, while the former motion is coupled with two tail vortices with designated strength and initial configurations, which are prescribed by the solution to the Föppl's problem. It is found that the configuration and the velocity of the tail vortices change due to the fixed cylinder, which in turns generates a hydrodynamic force affecting the approach of the moving cylinder.

11:09AM HT.00004 The dynamics of a solid-body rotation flow in a finite-length pipe[1] , SHIXIAO WANG, University of Auckland, ZVI RUSAK, Rensselaer Polytechnic Institute — The dynamics of a perturbed, incompressible, inviscid and axisymmetric solid-body rotation flow in a finite-length, straight, circular pipe is studied through a theoretical analysis of the steady-state equations and direct numerical simulations. The flow is subjected to non-periodic inlet and outlet conditions where the outlet flow is columnar. The computed bifurcation diagrams from both approaches agree as the pipe length increases. The results show the natural evolution of the flow at incoming swirl levels above critical into either a state with a breakdown (stagnation) zone, centred around the pipe axis, or into a state with a wall separation zone (where there is no axial and radial velocity). Each solution is a result of a different domain of attraction of initial perturbations. The work emphasizes that the solid-body rotation dynamics is dominated by the linear evolution of perturbations.

[1]Research is supported by U. Auckland Grant No. 3622940/9844.

11:22AM HT.00005 The dynamics of axisymmetric swirling flows in a diverging or contracting circular pipe[1] , ZVI RUSAK, Rensselaer Polytechnic Institute, SHIXIAO WANG, University of Auckland — This paper describes a study of the effect of pipe divergence and contraction on the stability and breakdown of axisymmetric swirling flows in a long, finite-length, circular pipe. The work extends the theory of Wang & Rusak (1997). The approach is based on a rigorous analysis of the axisymmetric, steady and inviscid flow equations with non-periodic boundary conditions. The analysis firmly establishes the global bifurcation of flow states in the pipe (solutions of the Squire-Long PDE) by relating it to the bifurcation of solutions of the columnar flow problem (solutions of the resulting ODE) and using a new flow force relationship between the inlet and outlet states. This technique provides a simple, yet exact, method of analyzing the complex flow behavior including transitions from near-columnar vortex states to flow fields with large separation (stagnation) zones along the pipe centerline (breakdown states) or along the pipe wall (swirl induced wall separation). Bifurcation diagrams for base vortex models including the solid- body rotation and the Burgers vortex are presented. The stability characteristics of the various branches of solutions and the flow dynamics in the pipe under various perturbations are discussed. Results show that pipe divergence or contraction significantly modify the global flow behavior in a straight pipe and shed light on the effect of pipe geometry on the mechanism of vortex breakdown.

[1]Research is supported by the University of Auckland, Grant No. 3622940/9844.

11:35AM HT.00006 Active Flow Enhancement for Heat Ejection in a Diverging Channel[1] , PABLO HIDALGO, ARI GLEZER, Georgia Institute of Technology — Small-scale motions that are induced within the core flow of a heated, high-aspect ratio diverging channel are investigated experimentally. The small scale motions are effected by the time-periodic vortex shedding from streamwise-embedded cantilevered reeds driven in resonance by integrated piezoelectric actuators and span the entire channel height. The induced vortices are advected with the core flow, disrupt the thermal boundary layers and result in a significant enhancement of the local heat transfer coefficient along the channel. Deliberate interactions between the reeds and a given core flow along secondary channels that are formed downstream of a center partition lead to controlled flow oscillations over a range of vibration frequencies. These interactions and the flow characteristics between the tip of the vibrating reed and the partition are investigated using high resolution particle image velocimetry (PIV). Of particular interest are the effects of variation in the reed motion and its distance to the partition on the induced small-scale motions and mixing and thus, on local and global heat transfer across the channel. Supported by DARPA and UTRC.

[1]Supported by DARPA and UTRC.

11:48AM HT.00007 Flow asymmetry and vortical structures behind a rotating tire , JOHN AXERIO, GIANLUCA IACCARINO, Stanford University — The wake behind both stationary and rotating tires is dominated by two strong counterrotating vortices. Experimental observations and numerical simulations have shown that one vortex may become stronger thus altering the wake dynamics and stability. The objective of this study is to investigate the main causes of near-wake and far-wake asymmetry. Factors such as tire camber angle, hub cavities, spoke geometry and the flow through the brake system, are all investigated as possible sources of asymmetry; it is shown that the most critical factor in flow asymmetry is the flow through the hub of the tire. When this is primarily in one direction (inboard to outboard side), the inboard vortex increases in size and intensity, eventually overwhelming the outboard vortex. We also study the transient features and timescales associated with the counterrotating vortex pair. Transient feature extraction is typically not possible with traditional techniques like PIV and RANS computer simulations. As a result, an LES simulation is used to extract eddy turnover timescales and vortex core trajectories for a rotating tire at Re=500,000.

12:01PM HT.00008 On point vortex equilibria in a bounded circular domain , GEORGES CHAMOUN, Technical University of Denmark, MARK STREMLER, Virginia Tech — We develop a general and systematic approach for solving point vortex equilibria in a bounded domain. The motivation for this work is that most, if not all, vortex equilibria in the physical world occur within a confined container filled with a finite amount of fluid. However, most of the existing models for vortex equilibria are in the unbounded plane. Furthermore, the few manuscripts that analyze vortex equilibria in the presence of a solid boundary present results for very simple configurations. We use the Hamiltonian point vortex model and the circle theorem for the equations of motion. We formulate the problem as one in linear algebra with the positions of the vortices given, and we use singular value decomposition to determine the vortex strengths necessary for relative equilibrium. We illustrate the feasibility of this technique with a few examples that are physically motivated from experiments, and we comment on linear stability.

12:14PM HT.00009 Swirling Strength Vortex Study in Confined Rectangular Jet , BO KONG, MICHAEL OLSEN, RODNEY FOX, JAMES HILL, Iowa State University — Vortex behavior in confined rectangular jet (Re = 20K, Re = 50K) were examined by using vortex swirling strength as a defining characteristic. Instantaneous velocity fields were collected for by using Particle Image Velocimetry(PIV). Swirling strength fields were calculated from velocity fields, and then filtered with a universal threshold of 1.5 times of swirling strength RMS value. By identifying clusters in filtered swirling strength fields, vortex structures were defined. Both instantaneous swirling strength field data and vortex population calculation indicate that the positively (counterclockwise) rotating vortices are dominant on the left side of the jet, and negatively (clockwise) rotating vortices are dominant on the right side. As flow develops further downstream, vortex population decreases and the flow approach channel flow. At the locations of the left peak of turbulent kinetic energy, two point spatial cross-correlation of swirling strength with velocity fluctuation were calculated. Linear stochastic estimation was also used to interpret the spatial correlation results and to determine conditional flow structures. High speed PIV data were also analyzed by using swirling strength technique to trace development of vortices. Vortex trajectories were found by tracing individual swirling strength clusters. The speed and strength of individual vortex were also studied by using this method.

12:27PM HT.00010 Numerical simulation of impurity propagation in sea channels , DMITRO CHERNIY, Kiev National Taras Shevchenko University, STANISLAV DOVGIY, ALEXANDRE GOURJII, Institute of Hydromechanics, National Academy of Science o Ukraine — Building the dike (2003) in Kerch channel (between Black and Azov seas) from Taman peninsula is an example of technological influence on the fluid flow and hydrological conditions in the channel. Increasing velocity flow by two times in a fairway region results in the appearance dangerous tendencies in hydrology of Kerch channel. A flow near the coastal edges generates large scale vortices, which move along the channel. A shipwreck (November 11, 2007) of tanker "Volganeft-139" in Kerch channel resulted in an ecological catastrophe in the indicated region. More than 1300 tons of petroleum appeared on the sea surface. Intensive vortices formed here involve part of the impurity region in own motion. Boundary of the impurity region is deformed, stretched and cover the center part of the channel. The adapted vortex singularity method for the impurity propagation in Kerch channel and analyze of the pollution propagation are the main goal of the report.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HU Granular V: Particles in Fluids and Interparticle Forces 200I

10:30AM HU.00001 Controlling cohesive forces in granular media , CHRISTOPH GÖGELEIN, MATTHIAS SCHRÖTER, MARTIN BRINKMANN, STEPHAN HERMINGHAUS, Max-Planck-Institut für Dynamik und Selbstorganisation, Bunsenstrasse 10, 37073 Göttingen, Germany — When adding a small amount of water to a pile of granular matter, e.g., sand heap, close-by grains can be connected by liquid bridges [1]. Thus, the material becomes plastically and can sustain a larger stress as compared to dry sand. Our general aim is to compare the mechanical properties of wet and dry granular media. For this purpose, we use a suspension of micrometer large glass or Latex spheres dispersed in a binary liquid mixture. The suspending water-lutidine(oil) mixture exhibits a lower critical solution temperature leading to a water-oil-like phase separation slightly above ambient temperature. Close to this demixing region, the oil-like phase undergoes a pre-wetting transition on the particle glass surface inducing liquid bridges [2]. Thus, by varying the temperature we can switch the liquid bridges on and off. We will report on our attempts to directly visualize the formation and control of liquid bridges using confocal and non- confocal microscopy.

[1] M. Scheel, *et al.*, Nature Materials 7, 174 (2008)
[2] D. Beysens, and D. Esteve, Phys. Rev. Lett. 54, 2123 (1985)

10:43AM HU.00002 Energy dissipation and clustering in granular streams , SCOTT WAITUKAITIS, JOHN ROYER, HELGE GRUETJEN, HEINRICH JAEGER, James Frank Institute, University of Chicago — The presence of weak cohesive forces between macroscopic grains can lead to the break up of a free falling granular stream, similar to the surface-tension-driven break up of a liquid stream[1]. This sensitivity to minute forces suggests that these free falling streams could serve as a tool to probe the interactions between grains. In order to investigate the connection between the stream dynamics and the grain-grain interactions, we perform molecular dynamics simulations of a granular stream freely falling out of a hopper varying the cohesion and inelasticity of the grains. We find that in the absence of cohesive forces the stream breaks apart into isolated grains, in contrast to the clustering observed in simulations of inelastic granular gases. For sufficiently high cohesive forces we reproduce the break up of stream into droplets, while with lower cohesive forces the stream breaks up into smaller clusters consisting of only a few grains. Measuring the change in contact number and decay of velocity fluctuations with depth, we characterize the different regions of the force-inelasticity phase space.

[1] Royer, J. R. et al. Nature **459** 1110 - 1113 (2009).

10:56AM HU.00003 A model for elastohydrodynamic collision of spheres in liquid , JOHN WELLS, Ritsumeikan Univ., J.T. JENKINS, Cornell Univ. — We pursue a reduced-order model of normal "elastohydrodynamic" collision between two spheres in incompressible liquid (Wells 1993; *Proc. Powders & Grains '93*, "W93"). W93 *assumed* a "quasi-Hertzian" surface depression profile, yielding the first reduced-order model ODEs for this problem. Trajectories and coefficients of restitution agreed well with simulations (Davis, Serayssol, & Hinch , 1986; *J. Fluid Mech* 201; "DSH86"). To avoid numerical integrations such as those in W93, Lian *et al.* (1996, *J. Fluid Mech* 311) assumed additionally that the gap inboard of a certain radius was constant, leaving only the centerline gap as an unknown. Then introducing an adjustable constant, they recovered coefficients of restitution close to DSH86. The present work extends W93 and Lian *et al.* to achieve, without adjustable constants, the analytical character that until now was only afforded by perturbation approaches (DSH86) wherein centerline deformation is assumed to be much smaller than the gap.

11:09AM HU.00004 The mechanism for shear thickening in suspensions , ERIC BROWN, HEINRICH JAEGER, University of Chicago — Densely packed suspensions can shear thicken, in which the viscosity increases with shear rate. Video microscopy along with rheology measurements show the shear thickening regime is a transition from negligible particle motion at low stresses to fully developed shear flow at higher stresses. The onset of shear thickening occurs when the shear stress is sufficient to pull particles apart; for example against gravity for large particles, and can be tuned by inducing electric or magnetic dipoles. Dilation can be observed as particles penetrate the fluid surface in the high stress regime. The maximum stress of the shear thickening regime is shown to match, for a wide range of suspensions, the ratio of surface tension divided by a radius of curvature comparable to the particle size. This suggests the large stress associated with shear thickening comes from capillary forces as a consequence of dilation.

11:22AM HU.00005 Liquid effect on a granular avalanche: experiments and discrete element simulation using a liquid-modified contact model , F.-L. YANG, ME, National Taiwan University, W.T. CHANG, C.S. CHEN, S.H. HSIEH, Y.T. HUANG, SOLID-LIQUID TWO-PHASE FLOW LAB TEAM, COMPUTER-AIDED ENGINEERING GROUP TEAM — This work extends a conventional Discrete-Element method to simulate the avalanche process and the subsequent bulk motion of a wet granular mixture, composed of identical solid spheres fully-immersed in a viscous liquid. A linear soft-sphere contact model is developed for interactions between dry surfaces to reproduce both the Hertzian contact time and the overall energy loss, characterized a measured coefficient of restitution. The contact model is modified for liquid dissipation effects according to previous experimental data. To assign parameters for tangential interactions, this works uses the measured bulk dynamic behavior, differing from most existing schemes. The simulated mixture motions are compared to experimental data, conducted on a lab-scale flume, at both flow and particle scales. General agreement is obtained on the whole granular avalanche process and the early stages of the subsequent downstream bulk motion. The validated simulation results are further employed to extrapolate bulk dynamics difficult to measure in the experiments, including 3D profiles of bulk velocity, strain, and granular temperature. The obtained results will be adopted to stress how liquid modifies the unsteady behavior of a granular mixture.

11:35AM HU.00006 Liquid effects on the dynamics of granular avalanche , YUNG-TA HUANG, FULING YANG, Department of Mechanical Engineering, National Taiwan University — This work examines experimentally how a viscous incompressible liquid modifies the behavior of a granular bulk in avalanche. A large and a small laboratory flume of identical length-to-width ratio have been constructed to initiate granular avalanche. Two types of monatomic spheres were used, preserving the ratio between the particle diameter and the flume width, with or without water to prepare the dry and the immersed mixtures. The bulk motion was monitored from the side by a high-speed digital camera. Quantitative measurements were conducted by integrating the techniques of particle-tracking velocimetry and coarse-grained spatial average. The examined bulk dynamics includes the two-dimensional profiles of bulk velocity and strain as well as the distribution of bulk solid volume fraction and granular temperature. The bulk slip velocity at the flume base and the volume discharge rate of a dry and an immersed mixture were also compared in space and in time throughout the avalanche. Further, a control volume analysis has been performed on the bulk mass, momentum, and energy (granular temperature) transport to extrapolate the stress fields. Finally, the scaling issue on granular flow dynamics will be addressed by comparing the bulk dynamics, on a non-dimensional reference frame, that was measured on the two flumes.

11:48AM HU.00007 Flow of a dense granular model suspension , CLAIRE BONNOIT, ANKE LINDNER, ERIC CLEMENT, PMMH-ESPCI — We experimentally study the flow of dense granular suspensions on an inclined plane. The model suspensions are made of mono-disperse, spherical, non-Brownian polystyrene beads immersed in a density-matched silicon oil. The volume fraction ϕ varies from 30% to 61%. During the flow on the inclined plane we measure simultaneously the surface velocity and the layer thickness. We identify two different scalings: one identical to flow of a viscous liquid at high thickness and one identical to flow of a dry granular media at low thickness. We show that particle migration can be neglected. In the viscous regime, this set-up thus allows for directly measuring the shear viscosity up to volume fractions of 61%, which is a challenge in a classical rheometer. We find that the inclined plane is a useful apparatus to explore the continuous transition from an effective viscous flow to a dense granular flow. As this geometry fixes the ratio of normal to tangential forces in the layer with respect to the angle, we are able to propose a rheological model based on a friction law.

12:01PM HU.00008 Ballistic to centrifuging flow of granular materials in rotating tumblers , RICHARD M. LUEPTOW, GABRIEL JUAREZ, PENGFEI CHEN, JULIO M. OTTINO, Northwestern University — The critical rotation speed for centrifuging of granular materials in horizontal rotating tumblers, determined by the Froude number, was studied experimentally and computationally as a function of the particle diameter, tumbler fill fraction, interstitial fluid, and tumbler length. Particle size does not affect the critical speed for centrifugation provided that the fill fraction is below 50%. The actual critical speed is typically 1.4 times the theoretical speed for centrifugation. Above 50% fill, the critical speed increases with particle size. Simulations indicate that even for the centrifuging condition there is relative motion of particles due to gravity, especially near the free surface. The presence of an interstitial fluid alters the critical speed due to buoyancy.

12:14PM HU.00009 Measurements of the Motion and Orientation of Rods in 2D Chaotic flow , SHIMA PARSA MOGHADDAM, GREG A. VOTH, Wesleyan University — We study the dynamics of rod particles in a two dimensional time-periodic flow driven by Lorentz force. Video particle tracking is used to make accurate measurements of the motion and orientation of fluorescent rods along with the fluid velocity field. When the rods are very short, their rotation rate is dictated by the local velocity gradients. We study the deviations from this simple model as the rod length increases. The rods partially align with the stretching direction defined by the eigenvectors of the Cauchy-Green strain tensor.

12:27PM HU.00010 Sedimenting spheres in bubbly fluid: a fluid Galton model , MICHAEL HIGLEY, ANDREW BELMONTE, Pennsylvania State University — A solid sphere sinking in a bubbly fluid and a solid sphere falling through a crowded bed of rigid obstacles (in air) share two common traits: the settling speed is slowed by the obstructions, and the sphere exhibits random lateral motion. In a previous study of sedimenting spheres in bubbly fluid we quantified both of these effects. Here we present a mathematical model which begins as an adaptation of Galton's board to the sedimenting sphere, which allows us to introduce various physical effects of the bubbly fluid, and test their importance, particularly bubble collisions. Comparison is made with experimental results.

Monday, November 23, 2009 10:30AM - 12:40PM –
Session HV Swimming III 205A-D

10:30AM HV.00001 A Fast-Starting Robotic Fish , YAHYA MODARRES-SADEGHI, MATTHEW WATTS, JOE CONTE, FRANZ HOVER, MICHAEL TRIANTAFYLLOU, MIT TOWING TANK TEAM — We have built a simple mechanical system to emulate the fast-start performance of fish. The system consisted of a thin metal beam covered by a urethane rubber fish body. The body form of the mechanical fish in this work was modeled from a pike species, which is the most successfully studied fast-start specialist species. The mechanical fish was held in curvature and hung in water by two restraining lines, which were simultaneously released by pneumatic cutting mechanisms. The potential energy in the beam was transferred into the fluid, thereby accelerating the fish, similar to a pike. We measured the resulting velocity and acceleration, as well as the efficiency of propulsion for the mechanical fish model and also ran a series of flow visualization tests to observe the resulting flow pattern. We also studied the influence of stiffness and geometry of the tail on the efficiency of propulsion and flow pattern. The hydrodynamic efficiency of the fish, calculated by the transfer of energy, was around 10%. Flow visualization of the mechanical fast-start wake was also analyzed, showing that the acceleration is associated with the fast movement of an intense vortex in a near-lateral direction.

10:43AM HV.00002 Biologically inspired impulsive starting and maneuvering for solitary and aggregate systems , ALEXANDRA TECHET, MIT — Fast starting and maneuvering in the aquatic realm typically involve the formation of distinct vortex rings that deliver an impulsive change in the animal's momentum. This enables these aquatic animals to maneuver in smaller spaces than that required by conventional underwater vehicles. PIV and dye visualization results from fast-starting and jumping fish, as well as impulsively starting flapping foils and propellers will be compared with unsteady propulsion by salps. Salps, or pelagic tunicates, are common gelatinous organisms in oceanic waters, which swim and maneuver by jet propulsion. Inspecting the wake generated by a rapidly maneuvering fish, foil or propeller offers insight into the impulse imparted on the system during the maneuver. Modeling the wake of maneuvering systems as a series of vortex ring impulses, with considerations taken for added mass effects, allows for relatively straightforward analysis. The swimming and maneuvering of aggregate swimmers, e.g. those chained together in series or parallel, can be modeled using a series of distinct vortex rings generated by each individual in the chain, with some phase shift between each individual.

10:56AM HV.00003 Vortex Formation and Acceleration of a Fish-Inspired Robot Performing Starts from Rest , ADAM DEVORIA, JONATHAN BAPST, MATTHEW RINGUETTE, The State University of New York at Buffalo — We investigate the unsteady flow of a fish-inspired robot executing starts from rest, with the objective of understanding the connection among the kinematics, vortex formation, and acceleration performance. Several fish perform "fast starts," where the body bends into a "C" or "S" shape while turning (phase I), followed by a straightening of the body and caudal fin and a linear acceleration (phase II). The resulting highly 3-D, unsteady vortex formation and its relationship to the acceleration are not well understood. The self-propelled robotic model contains motor-driven joints with programmable motion to emulate phase II of a simplified C-start. The experiments are conducted in a water tank, and the model is constrained to 1 direction along rails. The velocity is measured using digital particle image velocimetry (DPIV) in multiple planes. Vortex boundaries are identified using the finite-time Lyapunov exponent, then the unsteady vortex circulation is computed. The thrust is estimated from the identified vortices, and correlated with the circulation and model acceleration for different kinematics.

11:09AM HV.00004 3D Numerical simulations of the C-start of a Bluegill Sunfish[1] , VENKAT R.T. NARAYANAN, IMAN BORAZJANI, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, University of Minnesota, ERIC D. TYTELL, University of Maryland, College Park, GEORGE V. LAUDER, Harvard University — Obtaining the 3D flow field, forces, and power produced during the fast start maneuvers of fish is essential for studying this behavior from the hydrodynamics perspective. During a typical fast start, which is typically referred to as the C-start, the fish initially bends its body in a C shape manner and then with a fast stroke bends out of the C shape. We carry out high-resolution, 3D simulations of a bluegill sunfish performing a C-start maneuver. The body geometry and motion during the C-start are obtained from the experimental. We used high-speed video and particle image velocimetry to quantify body motion and flows produced during the C-start. We carry out simulations both with the entire motion prescribed and by prescribing only the deformation of the body but predicting the motion of the fish center of mass via a fluid-structure interaction approach. The computed results are compared with experimental observations and analyzed to further elucidate dynamics and three-dimensional structure of the C-start flowfields.

[1]This work was supported by NSF Grants 0625976 and EAR-0120914 and the Minnesota Supercomputing Institute.

11:22AM HV.00005 A wake-based correlate of swimming performance in seven jellyfish species , JOHN DABIRI, California Institute of Technology, SEAN COLIN, Roger Williams University, KAKANI KATIJA, California Institute of Technology, JOHN COSTELLO, Providence College — Animal-fluid interactions have been hypothesized as a principal selective pressure on the evolution of aquatic and aerial animals. However, attempts to discover the fluid dynamic mechanisms that dictate the fitness of an animal—or even to quantify 'fitness'—have been limited by an inability to measure the fluid interactions of freely moving animals (i.e., in the absence of tethers or artificial water/wind currents) in comparative studies of multiple species with similar evolutionary histories. We used digital particle image velocimetry (DPIV) measurements to calculate wake kinetic energy, drag, and swimming speed of the seven co-occurring species of free-swimming jellyfish. Using this new data, we demonstrate that the swimming and foraging behavior are related to a robust fluid dynamic threshold between two distinct configurations of the wake vortices. The transition between the two wake vortex configurations is known as optimal vortex formation, because it maximizes the fluid dynamic thrust generated for a given energy input (Krueger and Gharib, Phys. Fluids 2003). By comparing the observed wake structures created by each jellyfish species with the optimal vortex configuration, we are able to predict their relative swimming efficiencies and proficiencies and to deduce their corresponding ecological niches.

11:35AM HV.00006 Controlling pulsatile jet formation number with variable diameter exit nozzle for maximum impulse generation[1] , MIKE KRIEG, TYLER THOMAS, KAMRAN MOHSENI, University of Colorado — Both jellyfish and Squid propel themselves by ejecting high momentum vortex rings. A set of vortex ring generating thrusters were developed and tested for application in underwater vehicle propulsion. Vortex rings generated from a steady piston cylinder mechanism have a universal formation time, known as the formation number (Gharib et al. 1998), associated with reaching maximum circulation, where the vortex ring separates from its trailing shear flow. The non-dimensional jet formation time (also called the stroke ratio) plays a key role in the thrust output of the device; since thrusters operating above the formation number re-ingest the trailing jet. A variable diameter exit nozzle was used to increase the formation number of the jet to maximize thrust (which is a technique observed in squid and jellyfish locomotion). Visualization studies confirmed the ability to delay the onset of ring "pinch-off", using a variable nozzle, and the thrust was empirically shown to achieve a higher maximum. Additionally, a fluid slug model which was developed to predict the thrust output was adapted to incorporate a changing nozzle diameter. This model was verified with the empirical thrust data and was again shown to be accurate for stroke ratios below the formation number.

[1]The authors thank ONR for financial support.

11:48AM HV.00007 An Experimental Investigation of the Feeding Currents Generated by Upside-Down Jellyfish , ARVIND SANTHANAKRISHNAN, LAURA MILLER, UNC Chapel Hill — The flow characteristics of oblate medusan swimmers such as the moon jellyfish (*Aurelia*) have been examined to understand the bio-fluid mechanics of feeding via unsteady propulsion (see Dabiri *et al.*, J. Exp. Biol., 2005). The upside-down jellyfish (*Cassiopea*) differs from the commonly observed swimming forms of scyphomedusae in that it is naturally found adhered to the muddy bottoms of shallow ocean waters. While they swim when disturbed, these organisms prefer to otherwise attach their bell surface to the floor and direct their oral arms upwards. Prey capture is accomplished by pulsatile contractions of the bell. The flow generated by the unsteady pulsations is examined using a combination of DPIV and morphological measurements. The phase-averaged flow field closely resembles a blowing jet centered about the body, with fluid entrainment occurring near the bell surface. Reversed flow regions are identified during both the contraction and relaxation phases. The effect of changing bell diameter on the jet as well as the production of flow structures is investigated. A qualitative comparison of the flow field between these organisms and swimming medusae will be presented.

12:01PM HV.00008 Lagrangian coherent structures in jetting and paddling jellyfish swimming , DOUG LIPINSKI, KAMRAN MOHSENI, Univ. of Colorado — Lagrangian coherent structures (LCS) are a relatively new technique for visualizing and analysing structures and transport in complex fluid flows. In this study, we use LCS to examine the flow created by swimming jellyfish. We focus on identifying structures which contribute to feeding and propulsion and find several interesting results. Jellyfish which use different methods of propulsion create very different flow structures during swimming which are complimentary to the type of propulsion used. Additionally, we investigate the relationship between flow structures, pressure and swimming performance for the jetting Jellyfish, Sarsia tubulosa. We have previously detected structures within the bell of Sarsia tubulosa and we now focus on examining how these structures may impact the jellyfish's swimming.

12:14PM HV.00009 Pulsed Jet Propulsive Efficiency of a Small Underwater Vehicle at Low *Re* [1] , ALI MOSLEMI, PAUL KRUEGER, SMU, ALI MOSLEMI TEAM — Propulsive efficiency of steady jet propulsion decreases as Reynolds number (*Re*) decreases. Pulsed-jet propulsion can be an alternative option for traditional jet propulsion since it generates higher thrust than steady jet with the same mass flux rate. Moreover, previous work with a self-propelled pulsed-jet vehicle (Robosquid) at *Re* in the range 1300 – 2700 has shown: (a) an increase in pulsed jet efficiency at higher duty cycle (St_L) and lower jet pulse length-to-diameter ratios (L/D), and (b) the propulsive efficiency can be comparable to or exceed that for an equivalent steady jet when $L/D < 4$ and $St_L > 0.5$. A simple analysis suggests further propulsive efficiency gains for pulsed jets over steady jets will be realized as Re is reduced. To test this prediction, Robosquid will be tested in glycerin to achieve *Re* less than 1000. The effect of St_L and L/D on propulsive efficiency will be measured at these lower *Re* and compared with the previous results at higher *Re*.

[1]This work is supported by NSF grant CBET - 0347958.

12:27PM HV.00010 Stokesian Jellyfish , ARTHUR EVANS, SAVERIO SPAGNOLIE, ERIC LAUGA, University of California, San Diego — Most studies into the swimming of microscopic organisms are directed at flagellated cells, ciliated bodies, or other biologically relevant models. However, biological systems need not be emulated in order to produce locomotion. We present here a proof-of-principle model for a closed bilayer vesicle swimming at low Reynolds number due to a prescribed physically-actuated shape change of the surface. By modulating the volume and membrane composition of the vesicle via osmotic or colloidal effects, the vesicle shapes change continuously in thermal equilibrium, leading to non-reciprocal deformation, and therefore swimming, if the proper conditions are satisfied.

Monday, November 23, 2009 10:30AM - 12:14PM –
Session HW Swimming IV: Micro-organisms I 208A-D

10:30AM HW.00001 A microfluidic study of biofilms on topographically complex surfaces , WILLIAM DURHAM, ALBERTO LEOMBRUNI[1], MATTHEW MCKINLEY[2], ANNA SHCHERBINA[3], ROMAN STOCKER, MIT — A biofilm forms when bacteria attach to a surface and secrete sticky polymeric substances. Several factors control biofilm formation and maintenance, including cell motility, erosion by fluid shear, bacterial growth, nutrient diffusion, and surface properties. In particular, the surface is often topographically complex and allows heterogeneous microenvironments to develop. We studied how these processes influence biofilm dynamics using a patterned microfluidic channel, composed of an array of cylinders of random diameter and position. We tracked the evolution of a biofilm of fluorescently labeled *Escherichia coli* cells under flow over 48 hours and used periodic injections of microspheres to quantify the flow field. The biofilm first forms as a front that travels three orders of magnitude slower than the mean fluid velocity, then breaks into a series of patches separated by preferential flow channels. This striking channelization was rationalized using a simple network model. A full understanding of these dynamics will be relevant to problems in environmental and medical sciences.

[1]also affiliated with Politecnico di Milano
[2]Undergraduate Researcher
[3]Undergraduate Researcher

10:43AM HW.00002 ABSTRACT HAS BEEN MOVED TO AE.00010 –

10:56AM HW.00003 Investigating bacteria-surface interactions with microfluidics and Digital Holographic Microscopy , HARSH AGARWAL, University of Minnesota, MICHAEL BARRY, ROMAN STOCKER, Massachusetts Institute of Technology, JIAN SHENG, University of Minnesota — Quantitative data of swimming characteristics of bacteria in the shear flow adjacent to a surface are crucial for understanding cell attachment and detachment, and thus biofilm formation. We combined microfluidics and holography to expose *Escherichia coli* AW405 to a carefully controlled flow environment and visualize their movement in three dimensions. We investigated wall shear rates up to 200 (1/s) and recorded holograms at 40X magnification and 15fps for several minutes. Three-dimensional locations and orientations of bacteria were extracted from numerically reconstructed images. We obtained thousands of 3D trajectories over a sample volume of 380×380×200 μm, with a resolution of 0.2 μm in the two in-plane directions and 1 μm in the out-of-plane direction. Preliminary results revealed a range of behaviors, including circular trajectories near surfaces and migration normal to the wall. We expect that ongoing analysis will provide robust statistics of wall effects on bacterial motility. Sponsored by NIH (1-R21-EB008844-01) and NSF (CBET-0844647, DBI-0852875)

11:09AM HW.00004 Dynamics of Enhanced Tracer Diffusion in Suspensions of Swimming Microorganisms[1] , J.P. GOLLUB, J.S. GUASTO, Haverford Coll., K.C. LEPTOS, A.I. PESCI, R.E. GOLDSTEIN, U. Cambridge — We observe and statistically quantify the enhanced transport of passive tracer particles in suspensions of swimming microalgae, Chlamydomonas reinhardtii. These bi-flagellated, single-celled eukaryotes (10 μm diameter) swim with a breast-stroke motion of their flagella at speeds of about 100 μm/s and exhibit a heterogeneous trajectory shapes. Fluorescent tracer particles (2 μm diameter) allowed us to quantify the enhanced mixing caused by the swimmers, which is relevant to marine ecology. As the swimmer concentration increases, the probability density functions (PDFs) of tracer displacements develop strong exponential tails, and the Gaussian core broadens; the diffusivity grows linearly with concentration. For a given swimmer concentration, the displacement PDFs show self-similar behavior and diffusive scaling in time. High-speed imaging of tracer-swimmer interactions demonstrates the importance of flagellar beating in creating oscillatory flows that exceed Brownian motion out to about 5 cell radii from the swimmers.[2]

[1]Supported by NSF DMR-0803153, Leverhulme Trust, BBSRC, and Schlumberger Chair Fund.
[2]K.C. Leptos et al., submitted to Phys. Rev. Lett (2009)

11:22AM HW.00005 Bacteria swimming in shear , MARCOS, ROMAN STOCKER, Massachusetts Institute of Technology — The watery habitat of bacteria is typically in motion, exposing cells to fluid shear and thus to a viscous torque. By tracking individual bacteria exposed to a range of shear rates in microfluidic channels, we find that shear alters bacterial swimming patterns and in particular reduces movement across streamlines. This results from the bacteria undergoing Jeffery orbits, which bias cell orientation in the direction of the flow and hamper cross-streamline swimming. We speculate that this could significantly hinder chemotaxis and the quest for nutrients. A model based on resistive force theory is in good agreement with the observations. This process is a purely passive hydrodynamic effect. Further experiments suggest that bacteria do not actively respond to shear: nature has apparently not deemed it worthwhile to develop a shear sensor at the micrometer scale.

11:35AM HW.00006 Bacteria foraging in turbulent waters , JOHN TAYLOR, MIT, WENBO TANG, ASU, ROMAN STOCKER, MIT — Marine bacteria are the Ocean's recyclers, contributing to as much as 50% of the productivity of the marine food web. Bacteria forage on patches of dissolved nutrients using chemotaxis, the ability to swim up chemical gradients. As turbulence is ubiquitous in the Ocean, it is important to understand how turbulent flow conditions affect bacterial foraging. We used three-dimensional, isotropic direct numerical simulations coupled with a bacterial transport equation to address this problem. After the flow is continuously forced until it reaches a steady state, microscale nutrient patches are injected into the turbulent flow, and stirring produces thin nutrient filaments. Two populations of bacteria compete against each other: one population is motile and chemotactic ('active'), the other is non-motile ('passive'). The distribution of both populations is initially uniform. Chemotaxis allows active bacteria to cluster near the center of the nutrient filaments, increasing their nutrient uptake relative to passive bacteria. Increasing the turbulence intensity increases the short-term chemotactic advantage by quickly producing large gradients in the nutrient concentration, but also leads to rapid mixing of the nutrient field, which makes the chemotactic advantage short-lived. The results suggest that the evolutionary advantage of chemotaxis, based on the increase in nutrient uptake relative to the energetic cost of swimming, strongly depends on the turbulence level.

11:48AM HW.00007 Strain Variants in Swimming Characteristics of a Predatory Algae Species[1] , JIAN SHENG, University of Minnesota, JOSEPH KATZ, Johns Hopkins University, J. ADOLF, ALLEN PLACE — Digital holographic microscopic cinematography is used for measuring the 3D, time resolved, swimming behavior of toxic and non-toxic strains of the marine dinoflagellates *Karlodinium veneficum*. The experiments are performed in a 3×3 mm square cuvette at densities of ∼150,000 cells/ml, and holograms are recorded at 120fps and 20X magnification for 12-20s. In each case, we simultaneously track 200-500 cells in the 3mm deep sample, at a spatial resolution of 0.4×0.4×2 μm. Results show that all strains prefer vertical migration during phototrophic growth and localized foraging in response to prey. Strains capable of swimming in both left and right hand helices show stronger tendency towards vertical motion than right handed strains. Swimming-induced dispersion computed from Lagrangian trajectories corroborates the observed migration trends, and suggests a mechanism for predation-induced cell aggregation into dense bloom. Velocity spectra and conditional sampling of swimming modes will also be presented to elucidate locomotion of dinoflagellates.

[1]Sponsored by NSF.

12:01PM HW.00008 Fluid dynamics of phytoplankton with spines in unsteady shear flows , HOA NGUYEN, Tulane University, LEE KARP-BOSS, PETER JUMARS, University of Maine, LISA FAUCI, Tulane University — Spines and other thin projections from cell surfaces literally expand the volume of fluid with which a cell interacts and may provide effective levers on which the flow can act. We use an immersed boundary formulation to solve the coupled phytoplankton-fluid system to predict the 3D trajectories of the cells within a background flow. We examine the effect of spines on the trajectories, along with the effect of stiffness properties of these spines.

Monday, November 23, 2009 2:00PM - 2:35PM –
Session J1 Invited Session: Tumor Blood Vessel Dynamics 205A-D

2:00PM J1.00001 Tumor Blood Vessel Dynamics , LANCE MUNN, Harvard University — "Normalization" of tumor blood vessels has shown promise to improve the efficacy of chemotherapeutics. In theory, anti-angiogenic drugs targeting endothelial VEGF signaling can improve vessel network structure and function, enhancing the transport of subsequent cytotoxic drugs to cancer cells. In practice, the effects are unpredictable, with varying levels of success. The predominant effects of anti-VEGF therapies are decreased vessel leakiness (hydraulic conductivity), decreased vessel diameters and pruning of the immature vessel network. It is thought that each of these can influence perfusion of the vessel network, inducing flow in regions that were previously sluggish or stagnant. Unfortunately, when anti-VEGF therapies affect vessel structure and function, the changes are dynamic and overlapping in time, and it has been difficult to identify a consistent and predictable normalization "window" during which perfusion and subsequent drug delivery is optimal. This is largely due to the non-linearity in the system, and the inability to distinguish the effects of decreased vessel leakiness from those due to network structural changes in clinical trials or animal studies. We have developed a mathematical model to calculate blood flow in complex tumor networks imaged by two-photon microscopy. The model incorporates the necessary and sufficient components for addressing the problem of normalization of tumor vasculature: i) lattice-Boltzmann calculations of the full flow field within the vasculature and within the tissue, ii) diffusion and convection of soluble species such as oxygen or drugs within vessels and the tissue domain, iii) distinct and spatially-resolved vessel hydraulic conductivities and permeabilities for each species, iv) erythrocyte particles advecting in the flow and delivering oxygen with real oxygen release kinetics, v) shear stress-mediated vascular remodeling. This model, guided by multi-parameter intravital imaging of tumor vessel structure and function, provides a tool for identifying the structural and functional determinants of tumor vessel normalization.

Monday, November 23, 2009 2:00PM - 2:35PM –
Session J2 Invited Session: The Fluid Mechanics Inside a Volcano 208A-D

2:00PM J2.00001 The Fluid Mechanics inside a volcano , MICHAEL MANGA, University of California, Berkeley — The style and evolution of volcanic eruptions is dictated by the fluid mechanics that governs ascent of magma. Ascent in turn is influenced by the nucleation and growth of bubbles which provide the driving force for eruptions, the loss of gas from the magma, crystallization, fragmentation, and magma rheology. All such changes in the magma are coupled. We integrate models of all these processes to show when each dominates and how they interact within a conduit. In particular, we show that ascent rate controls eruptive behavior: slowly ascending magmas erupt effusively and rapidly ascending magmas erupt explosively.

Monday, November 23, 2009 2:40PM - 3:15PM –
Session K1 Invited Session: Unsteadiness of Shock Wave / Boundary Layer Interactions 205A-D

2:40PM K1.00001 Unsteadiness of Shock Wave / Boundary Layer Interactions[1] , NOEL CLEMENS, The University of Texas at Austin — Shock wave / boundary layer interactions are an important feature of high-speed flows that occur in a wide range of practical configurations including aircraft control surfaces, inlets, missile base flows, nozzles, and rotating machinery. These interactions are often associated with severe boundary layer separation, which is highly unsteady, and exhibits high fluctuating pressure and heat loads. The unsteady motions are characterized by a wide range of frequencies, including low-frequency motions that are about two orders of magnitude lower than those that characterize the upstream boundary layer. It is these low-frequency motions that are of most interest because they have been the most difficult to explain and model. Despite significant work over the past few decades, the source of the low-frequency motions remains a topic of intense debate. Owing to a flurry of activity over the past decade on this single topic we are close to developing a comprehensive understanding of the low-frequency unsteadiness. For example, recent work in our laboratory and others suggests that the driving mechanism is related to low-frequency fluctuations in the upstream boundary layer. However, several recent studies suggest the dominant mechanism is an intrinsic instability of the separated flow. Here we attempt to reconcile these views by arguing that the low-frequency unsteadiness is driven by *both* upstream and downstream processes, but the relative importance of each mechanism depends on the strength (or length-scale) of separation. In cases where the separation bubble is relatively small, then the flow is intermittently separated, and there exists a strong correlation between upstream velocity fluctuations and the separation bubble dynamics. It appears that superstructures in the upstream boundary layer can play an important role in driving the unsteadiness for this case. It is not clear, however, if the upstream fluctuations directly move the separation point or indirectly couple to a global instability. In cases where the separation is strong (and the bubble large) then the bubble pulsates owing to a global instability, as has been suggested by other researchers. In this case upstream turbulence may serve mainly as a source of broadband fluctuations that seed the large-scale instability of the separated flow.

[1]The author's work on this topic has been sponsored, over many years, by AFOSR and ARO. This support is gratefully acknowledged.

Monday, November 23, 2009 2:40PM - 3:15PM –
Session K2 Invited Session: Cavitation Inception and Bubble Dynamics in Vortical Flows 208A-D

2:40PM K2.00001 Cavitation Inception and Bubble Dynamics in Vortical Flows , STEVEN L. CECCIO, University of Michigan — The liquid is the core of a vortex can be significantly lower that the surrounding pressure, and possibly in tension. Small bubbles (nuclei) exposed to this tension can rapidly enlarge to fill the radial extent of the vortex core and then grow along the vortex axis. Such vortex cavitation can readily occur in the shed vortices of lifting surfaces or in turbulent shear flow such as jets and wakes. Incipient and developed vortex cavitation bubbles can exhibit complex dynamics as the bubble interacts with the surrounding flow. As the bubble changes volume within the vortex core, the vorticity distribution of the surrounding flow is modified, which then changes the pressures at the bubble interface. This coupling can produce volume oscillations with a period of the order of the vortex time scale, $\tau_V = 2\pi r_c/u_{\theta,max}$, where r_c is the vortex core radius and $u_{\theta,max}$ is its maximum tangential velocity of the vortex. However, the volume oscillation amplitude and frequency are quite sensitive to variations in the vortex properties, the rate and magnitude of the local pressure core pressure, and the nuclei's critical pressure. The axial and radial growth of elongated cavitation bubbles is also strongly coupled, especially near the axial extents of the bubble. Such complex growth, oscillation, and collapse of vortex cavitation bubbles can lead to both broadband and tonal sound emissions.

3:15PM - 3:15PM –
Session L1 Poster Session (15:15 - 17:00) Ballroom AB

L1.00001 Visualization of airflow in the wake of a ship superstructure[1] , C.J. BROWNELL, W.P. STILLMAN, J.H. GOLDEN, S.A. SIMPSON, L. LUZNIK, D.S. MIKLOSOVIC, G. WHITE, J.S. BURKS, M.R. SNYDER, U.S. Naval Academy — Helicopter landings on naval surface ships, such as cruisers and destroyers, must take place in the presence of an air wake created by flow over the ship superstructure. Wake turbulence over the flight deck makes piloted landings dangerous and difficult, and poses significant problems for the use of unmanned rotorcraft. To address this problem, a comprehensive set of experimental and simulation data are being collected via concurrent field tests, wind tunnel measurements, and CFD simulations. These data will facilitate an understanding of the wake turbulence produced under a variety of weather conditions, and will allow assessment of the fidelity of lower order flowfield estimates. A U.S. Navy Auxiliary Patrol (YP) Craft is used as a representative ship platform. The YP is over 100 ft long, has a similar shape to a modern destroyer, and has been modified to include a flight deck and hangar-like superstructure. Presented here are preliminary CFD results along with results from a large-scale flow visualization experiment. Qualitative information gleaned from the flow visualization is being used in the experimental design of upcoming quantitative air velocity measurements.

[1]Supported by ONR YIP

L1.00002 Numerical simulations of bell contractions of upside down jellyfish using the immersed boundary method , CHRISTINA HAMLET, LAURA MILLER, University of North Carolina-Chapel Hill — Pulsatile jet propulsion is one of the simplest forms of locomotion utilized by macroscopic organisms. Jellyfish use contractions of their bells to form vortex rings that facilitate feeding and locomotion. Once they grow to about 2 cm in diameter, the upside-down jellyfish (genus Cassiopea) situates itself on the ocean substratum with its oral arms towards the sun. These organisms primarily utilize zooanthellae for photosynthetic feeding, while also pulsing their bells to generate feeding currents and to move short distances. Preliminary numerical simulations are presented here which model the motion of the jellyfish as they pulse on the ocean floor. The motion of the bell is measured and fit to a mathematical model using video. The bell motion is used as an input in numerical simulations. Ultimately, contraction of muscle fibers which induce bell contractions will be simulated across Reynolds numbers using the immersed boundary method.

L1.00003 Experimental study of squeeze-film flow related to human voice production[1] , DAN LO FORTE, SCOTT THOMSON, Brigham Young University — Airway surface liquid (ASL) consists of a Newtonian/non-Newtonian bi-layer and lines the human airway, including the vocal folds. Several studies indicate that ASL properties affect vocal fold operation. In particular, adverse ASL properties may lead to voice discomfort and damage. This presentation will focus on research performed to elucidate the role of the ASL in voice production. Experimental measurements of squeeze-flow between parallel plates (similar to the ASL during vocalization) are presented. Simultaneous film thickness, force, and high-speed image data are reported and compared with computational solutions. The resulting squeeze force is compared for varying parameters including film thickness and fluid properties. This data will help develop an accurate ASL model to study its function in depth. Long-term results may help in treatment of voice disorders. Additionally this work provides experimental data to support the validation of existing thin-film squeeze-flow equations.

[1]This project was supported by Grant Number R03DC008200 from the National Institute on Deafness and Other Communicative Disorders (NIDCD).

L1.00004 ABSTRACT WITHDRAWN –

L1.00005 Computational Analysis of Human Blood Flow , YOGENDRA PANTA, HAZEL MARIE, MARK HARVEY, Youngstown State University — Fluid flow modeling with commercially available computational fluid dynamics (CFD) software is widely used to visualize and predict physical phenomena related to various biological systems. In this presentation, a typical human aorta model was analyzed assuming the blood flow as laminar with complaint cardiac muscle wall boundaries. FLUENT, a commercially available finite volume software, coupled with Solidworks, a modeling software, was employed for the preprocessing, simulation and postprocessing of all the models.The analysis mainly consists of a fluid-dynamics analysis including a calculation of the velocity field and pressure distribution in the blood and a mechanical analysis of the deformation of the tissue and artery in terms of wall shear stress. A number of other models e.g. T branches, angle shaped were previously analyzed and compared their results for consistency for similar boundary conditions. The velocities, pressures and wall shear stress distributions achieved in all models were as expected given the similar boundary conditions. The three dimensional time dependent analysis of blood flow accounting the effect of body forces with a complaint boundary was also performed.

L1.00006 Analysis of Signal Propagation in an Elastic-Tube Flow Model , SCOTT WAGGY, University of Colorado at Boulder, OZGUR AKMAN, The University of Edinburgh, SEDAT BIRINGEN, University of Colorado at Boulder — We combine linear and nonlinear signal analysis techniques to investigate the transmission of pressure signals along a one-dimensional model of fluid flow in an elastic tube. We derive a simple measure for the robustness of a simulated vessel against in vivo fluctuations in the pressure, based on quantifying the degree of synchronization between proximal and distal pressure pulses. The practical use of this measure will be in its application to simulated pulses generated in response to a stochastic forcing term mimicking biological variations of root pressure in arterial blood flow. Using spectral analysis methods based on synchronization theory, we introduce a novel nonlinear index for measuring the robustness of the model against fluctuations in the forcing signal, based on a general scheme for deriving low-dimensional measures of (biological) performance from higher-dimensional systems of equations.

L1.00007 Shape oscillation of bubbles in the acoustic field[1] , KEISHI MATSUMOTO, Div. Mechanical Engineering, Graduate Sch. Science & Technology, Tokyo University of Science, ICHIRO UENO, Dept. Mechanical Engineering, Fac. Science & Technology, Tokyo University of Science — We focused on dynamics of multiple air bubbles exposed to ultrasonic wave while ascending in water. The bubbles were injected into the pool filled with water from a vertical capillary tube, and then the ultrasonic wave of 20 kHz was applied from above toward the bubbles. Volume and shape oscillations of the bubbles were captured by a high-speed camera at frame rates up to 40000 fps with a back-lighting system. We realized three major phenomena in this experiment; excitations of surface wave, volume oscillation, and shape oscillation. In the present study we paid our special attention to the shape oscillation and the transition from the volume to the shape oscillations of the bubbles in a row. Effects of the bubble volume and the distance between the bubbles were examined. We got relation among distinct mode number n, which is order of harmonic, bubble diameter, and distance of the bubbles.

[1]A part of this study is partly supported by TEPCO Research Foundation.

L1.00008 On the effects of Schmidt number and particle settling velocity on the calculated sediment concentration profile , ALI KHOSRONEJAD, Post Doctoral Associate, St. Anthony Falls Lab. — The numerical simulation of natural stream flows is a powerful tool which is generally used to achieving the following goals: tracing the contaminants in natural waterways, computing the suspended sediment concentration, computing the bed evaluation of waterways, etc. In most natural rivers, the flow is turbulence and therefore the turbulent diffusion plays a critical rule in scalar term dispersion. To evaluate the effect of turbulent diffusion, the Schmidt number is widely used. In this paper, the effect of different Schmidt number on the calculated sediment concentration profiles is investigated. Also, the effect o sediment particle settling velocity of the computed sediment concentration profile has been investigated. Some standard test cases including net entrainment from a mobile bed, net deposition to the bed, dye material advection dispersion, have been chosen. For the net deposition, net entrainment and dye advection-dispersion the experimental data of Jobson and Sayre (1970), van Rijn (1981) and Jobson and Sayre (1970), respectively, has been used to comparing the model results by Schmidt numbers and settling velocity values. As the results, the most appropriate parameters for which a better prediction is achieved are presented for each case.

L1.00009 A Numerical Model for Time-Dependent Gravity-Driven Flow in a Collapsible Tube , AMANDA PETERS, MICHELLE BORKIN, SHREYAS MANDRE, Harvard University — We present details of a Navier-Stokes solver to address fluid flows through a circular tube with elastic walls. This is the first implementation of a large structured-grid fluid dynamics code on this architecture. This class of problems, fluid flow through collapsible tubes, is very important to the study of biological systems (respiratory system, circulatory system, etc.) and physical systems (fluid dynamics, engineering, etc.). In contrast to other models, we focus on integrating wall elasticity and time dependance. We successfully model the flow of blood through the jugular vein of a giraffe over time by numerically evaluating a series of hyperbolic PDEs using Lax-Wendroff. Through careful error and stability analysis, we were able to create an accurate and stable simulation. We were able to examine the role that elasticity plays at various length scales and determine it has an impact on the flow velocity over large length scales (i.e. a giraffe) whereas it is negligible over small length scales (i.e. a human) as it is likely overwhelmed by factors such as lateral flow and viscosity. This work presents a strong framework for future CFD studies regarding various human blood flow physiologies including the abdominal aorta.

L1.00010 Turbulence measurements in air near a heated vertical cylinder at high Raleigh number , ABOLFAZL SHIRI, Chalmers University of Technology, Gothenburg, Sweden, WILLIAM K. GEORGE[1], CNRS, LML, Lille, France — Measurements are presented up to 4 m height of the temperature and velocity statistics along a heated vertical cylinder of diameter 0.15 m. The cylinder was heated by water flowing through it, and mounted inside a 1.2 m container through which the ambient flow could be controlled. The difference between the wall and ambient temperatures was approximately 40 degrees C resulting in a maximum Rayleigh number based on length, $Ra = g\beta\Delta T x^4/\alpha\nu \approx 2.3 \times 10^{11}$. The velocity was obtained using two-component burst-mode LDA, while the temperature was measured using 1-micron platinum wires. Arrays of thermocouples were used to monitor the ambient and wall conditions. Particular attention has been given to the buoyancy and moment integral equations in order to evaluate the residual effects of stratification and co-flow.

[1]Visiting Professor

L1.00011 Measurement of concentration filed in miscible displacement in a thin gap by means of microelectrode , YUICHIRO NAGATSU, TAKASHI OGAWA, YOSHIHITO KATO, YUTAKA TADA, Nagoya Institute of Technology, Japan — When a miscible less-viscous liquid displaces a more-viscous one in a Hele-Shaw cell (a thin gap between two parallel plates, normally the gap width is less than 1 mm), it is known that a thickness of the less-viscous liquid's layer is expected to abruptly become small in a relatively long distance near the displacement front under a certain condition. We call this a sheet structure. The objective of the present study is to experimentally measure concentration profiles in the cell's gap direction to examine the expectation. However, because of the minuteness of the measured space, it is difficult to measure it by existing measurement techniques. In the present study, therefore, we attempt to measure it by means of a microelectrode which has been frequently used in the biological field. The obtained results suggest the presence of the sheet structure. The obtained concentration profile, however, shows that the sheet structure exists mainly in the upper half of the gap although we have been expected that the sheet structure is present in the middle of the gap.

L1.00012 Using stabilized finite elements for understanding the performance of organic solar cells , HARI KRISHNA KODALI, BASKAR GANAPATHYSUBRAMANIAN, Iowa State University — Organic solar cells (OSCs) fabricated from polymer blends are a promising alternative to inorganic photovoltaics. Computational modeling of OSCs is of significant utility towards understanding the relationship between topology/ morphology and efficiency. Simulation of OSCs requires determination of electric potential and electron/ hole densities, which is described by Boltzmann Transport Equation (BTE). As the direct solution of BTE is computationally challenging, the drift-diffusion model is used for modeling traditional semiconductor devices. The drift component in the drift-diffusion equations causes large convection instabilities. This is mitigated by 'Streamline Upwind Petrov Galerkin' (SUPG) stabilization traditionally used in convection dominated flow problems in computational fluid dynamics. Although this problem has been addressed for silicon semiconductors, simulation of organic bulk heterojunction (BHJ) solar cells presents new challenges. A comparative study is undertaken for solution of stabilized drift-diffusion equations with primitive, Slotboom and Quasi Fermi Level variables. A simplified drift-diffusion model (with analytical solution) is used to verify these implementations. The framework is utilized to investigate the effect of topology on photocurrent.

L1.00013 Analysis of wake structure downstream of rough surface using volumetric 3-D measurement , WING LAI, TSI Fluid Mechanics Group, JOSEPH SHAKAL, U of Wisconsin River Fall — Wake structures downstream of rough surface were analyzed by a 3-D volumetric image based measurement system. The wake structure was shown to have a direct correlation to the roughness of the surface, with the wake deviate from the boundary layer drastically for a very rough surface. A number of rough surfaces were tested to show the various wake formations due to various degrees of roughness. The 3-D volumetric system was employed for the measurement to allow the complete 3-D wake structure to be measured. Such 3-D results allowed instantaneous understanding of the roughness effect to the surrounding flows.

L1.00014 Observations of the turbulence kinetic energy dissipation rate in the mid-water column of an estuary , LUKSA LUZNIK, Department of Mechanical Engineering, US Naval Academy, LOUISE WALLENDORF, Hydromechanics Laboratory, US Naval Academy — A substantial number of oceanic observational studies show that beneath surface waves the turbulence kinetic energy dissipation rate is often much larger than in a comparable turbulent boundary layer over a rigid wall. However, the vertical extent of the region of enhanced dissipation rate is not well characterized primarily due to variability in atmospheric forcing and wave conditions. Here, observations of turbulent dissipation rate are examined together with simultaneous measurements of surface wind waves and tidal currents in Chesapeake Bay during the summer of 2009 under low to moderate wind conditions. The data were collected with two vertically separated acoustic Doppler velocimeters (ADVs) and a high-resolution pulse coherent profiler (Aquadopp) covering a range of depths from 2-4m below the surface in 5m of water. A bottom mounted acoustic wave and current meter (AWAC) provided wave measurements and mean velocity profiles. Estimates of dissipation rates are obtained from inertial subrange spectra derived from temporal (ADV's) and spatial (Aquadopp) data. Various dissipation scaling will be discussed to determine the relative importance of encountered wave and tidal conditions.

L1.00015 Modeling Subsurface Flows Driven by Planetary Libration , MICHAEL CALKINS, JEROME NOIR, JEFF ELDREDGE, JONATHAN AURNOU, University of California, Los Angeles — Longitudinal libration is a non-zero mean, time-periodic variation in a planetary body's rotation rate. The outermost solid shells of numerous planetary bodies in the solar system —including Mercury, Europa, and Earth's moon— are currently librating. This libration is capable of driving flows in these planets' liquid metallic cores and subsurface oceans through viscous, topographic or electromagnetic couplings. We have carried out a suite of laboratory and numerical hydrodynamic libration experiments in sphere and spherical shell geometries. This set-up allows us to focus, at present, on the purely viscous coupling problem. Laboratory experiments demonstrate that longitudinal libration is capable of generating time-periodic centrifugal instabilities near the librating solid boundary, as well as inertial waves and zonal flows in the fluid interior. In an effort to apply these results to librating planets, we have carried out axisymmetric numerical simulations that access more extreme parameter values than can be reached in the laboratory experiment. These simulations show that the nonlinear interaction of inertial waves is the primary mechanism responsible for the zonal flow generation, whereas the Reynolds stresses generated from centrifugal instabilities only weakly influence zonal flow strength.

L1.00016 Jamming ang energy propagation through dense granular matter[1] , XIAONI FANG, LOU KONDIC, NJIT — In previous work (Phys. Rev. E **79**, 041304 (2009)) we found using discrete element simulations that a reasonable description of energy propagation through dense (jammed) granular system in which volume fraction is keep fixed (CV protocol) can be reached by a linear wave equation with damping. In the present work we consider the systems where we either decrease the volume fraction, or a system under constant applied pressure (CP protocal). In both of the considered scenarios one may distinguish between the jammed and unjammed configurations, defined by the coordination number. We discuss how the nature of energy propagation changes as one goes through the jamming transition.

[1]Supported by NSF grants DMS-0605857 and DMS-0835611.

L1.00017 Jamming in Microfluidic Channels , CARLOS ORTIZ, KAREN DANIELS, ROBERT RIEHN, North Carolina State University — We experimentally investigate the flow of a colloid through a microfluidic device. The glass microfluidic device consists of a wide channel with spatially periodic funnels manufactured with photolithographic methods. The device was etched to a depth of about 1 micron that restricts the solid phase of the colloid, fluorescent polystyrene spheres with sub-micron radii, to quasi-2D motion. The liquid phase of the colloid is an aqueous solution with trace amounts of a non-ionic surfactant and with a pH about 2 units above the pKa of the surface groups on the polystyrene spheres to maintain a stable colloid at concentrations high enough to produce jamming. The flow rate of the colloid is controlled by a computer interfaced syringe pump with two controllable modes of operation: a continuous, steady mode that provides a plug-like velocity profile and a discrete, jerky mode that sends compressional waves of specifiable sizes through the colloid. Using fluorescence microscopy, we observe the interactions between the colloid and the glass funnels and investigate how the interaction depends on the funnel geometry. In particular, we investigate the jamming transition from a liquid-like flowing state to a solid-like stationary state.

L1.00018 DEM simulation of granular flow in a Couette device[1] , VIDYAPATI VIDYAPATI, Iowa State University, M. KHERIPOUR LANGRUDI, GABRIEL TARDOS, The City College of the City University of New York, JIN SUN, SANKARAN SUNDARESAN, Princeton University, SHANKAR SUBRAMANIAM, Iowa State University — We study the shear motion of granular material in an annular shear cell operated in batch and continuous modes. In order to quantitatively simulate shear behavior of granular material composed of spherical shaped grains, a 3D discrete element method (DEM) is used. The ultimate goal of the present work is to compare DEM results for the normal and shear stresses in stationary and moving granular beds confined in Couette device with experimental results. The DEM captures the experimental observation of transition behavior from quasi– static (in batch mode operation) to rapid flow (in continuous mode operation) regime of granular flows. Although there are quantitative differences between DEM model predictions and experiments, the qualitative features are nicely reproduced. It is observed (both in experiments and in simulations) that the intermediate regime is broad enough to require a critical assessment of continuum models for granular flows.

[1]U.S Department of Energy, National Energy Technology Laboratory

L1.00019 Large Reynolds number streak description using RNS , JUAN MARTIN, CARLOS MARTEL, ETSI Aeronauticos, Universidad Politecnica de Madrid — It has been recently shown [Choi, Nature, April 06 - Cossu et al., PRL, February 06] that the inclusion of 3D streaky structures in the boundary layer can make it remain laminar longer than the purely 2D Blasius flow. We compute the development of 3D streaks in the boundary layer over a flat plate using the so-called Reduced Navier Stokes equations (RNS). The RNS are a boundary layer like formulation, which is derived from the Navier-Stokes equations making use of the fact that in the large Re limit two very different spatial scales are present: one long (streamwise direction) and two short (spanwise and wall-normal direction). The resulting RNS are a nonlinear, parabolic, Re independent system that describes the streak structure in the large Re limit. The RNS streak computations are also much more less CPU costly than DNS for high Re, and they don't have the numerical problems that the PSE formulation exhibits (divergence of the results for small Δx, or blow-up of the solution when the amplitude of the deviation from Blasius is not small). In this work we comment the details of the numerical integration of the RNS, and we present some comparisons of the RNS results with the linear computations of streak development together with some fully nonlinear computations of streak evolution.

L1.00020 Instability of Thin Viscous Sheets , ANDREY FILIPPOV, ZHEMING ZHENG, Corning Incorporated — Thin viscous fluid flows are characterized by a small aspect ratio, i.e., the characteristic thickness is much smaller than the characteristic tangential length scale. They have been used in a variety of manufacturing applications, including curtain coating, film blowing, film casting, extrusion and optical fiber drawing. An asymptotic theory for predicting the thickness distribution and the geometry boundaries of a thin nearly planar fluid sheet and analyzing the stability of its shape is developed, where the sheet thickness and out of plane displacement are additional distributed variables along with the continuum velocities. The sheet motion in the transverse direction is described by a transient second order PDE expressing the normal momentum balance: Its analysis showed that the existence of compressive stresses inevitably leads to viscous sheet shape instability, while the corresponding stationary equation becomes of mixed type. Two practical examples involving viscous sheets have been considered: an isoviscous two-dimensional viscous sheet, which shape is unstable in the case when the compressive stress is applied by setting an obstacle to the flow at the exit end, and a 3D problem of viscous sheet redraw (constant stretching), where existence of compressive stresses leads to development of hyperbolic zones in the sheet, and the sheet shape instability develops.

L1.00021 Instability arisen on liquid jet penetrated in flowing liquid bath[1] , NAOTO OKA, Div. Mechanical Engineering, Graduate Sch. Science & Techanical, 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 penetration process and an instability on a liquid jet impinged to a flowing liquid pool. The impinged jet penetrates into the flowing bath accompanying with an entrainment of the ambient immiscible gas without coalescing with the liquid in the pool until the air wrap around the jet collapses. The wrapping air controls instabilities arisen on the jet. We observe the dynamic behaviors of the penetrated jet and the departure of the bubble of the wrapping gas at the tip of the collapsing jet by use of a high-speed camera in order to categorize the behaviors as functions of the velocities of the jet and flow in the pool. We also evaluate an averaged thickness of the wrapping gas through the observation.

[1]A part of this study is partly supported by TEPCO Research Foundation.

L1.00022 Wave-vortex interaction , CLAUDIO FALCON, Departamento de Fisica, Facultad de Ciencias Fisicas y Matematicas, Universidad de Chile, Casilla 487-3, Santiago, Chile, STEPHAN FAUVE, Laboratoire de Physique Statistique, Ecole Normale Superieure, CNRS , 24, rue Lhomond, 75 005 Paris, France — We present an experimental study on the effect of a electromagneticaly generated vortex flow on parametrically amplified waves at the surface of a fluid. The underlying vortex flow, generated by a periodic Lorentz force, creates spatio-temporal fluctuations that interact nonlinearly with the standing surface waves. We characterize the bifurcation diagram and measure the power spectrum density (PSD) of the local surface wave amplitude. We show that the parametric instability threshold increases with increasing intensity of the vortex flow.

L1.00023 Modeling of Flow and Water Quality Processes with Finite Volume Method due to Spreading and Dispersion of Petrochemical Pollution in the Hydro-Environments , EHSAN SARHADI ZADEH, KOUROSH HEJAZI — Having two water frontiers, namely (everlasting) Persian Gulf and Oman Sea in the south and Caspian Sea in the north, intense dependence on extracting and exporting oil, especially via marine fleets and ever-increasing development of petrochemical industry, Iran is exposed to severe environmental damages caused by oil and petrochemical industries. This essay investigates how oil spill is diffused and its environmental pollution is spread. The movement of oil spill, and its diffusion in water and its effects on water and the environment has been simulated by developing a Depth-Averaged numerical model and using the Finite Volume method. The existing models are not efficient enough to fulfill current modeling needs. The developed model uses the parameters useful in the advection and diffusion of oil pollutions in a model appropriate for predicting the transport of oil spill. Since the Navier-Stokes Equations play an important role in the advection and diffusion of oil pollutions, it is highly important to choose an appropriate numerical method in the advection and diffusion section. In this essay, choosing the methods used in the advection and diffusion have been emphasized and highly-accurate algorithms has been used in the advection terms. These algorithms are not present in similar models. The resulting equations have been solved using the ADI method. This method solves the unknown parameters with solving a Penta-Diagonal matrix in each time step. It does so without sacrificing the desired precision.

L1.00024 The refraction phenomena in the shock wave dispersion on plasma inhomogenities , ANNA MARKHOTOK, MMA, SVETOZAR POPOVIC, ODU — Recently we introduced a new mechanism of the shock wave (SW) dispersion on a boundary [1] and then employed it to control the structure of a SW front as it interacts with plasma. Now we consider same effect but under specific conditions on the interface with weak gradients at the interface, but stronger in the bulk. These conditions appear more realistic for most applications, and more important, the SW dispersion in this case becomes stronger. We derive all relations using the approach similar to [1]. Then the model applied to calculate the structure of the SW front dispersed on a plasma sphere numerically and compared with the existing experiments. The strength of the effect is demonstrated by comparing results of calculation for different conditions on the interface. Comparative calculations show significant difference in the shock front shapes and they are in complete agreement with the experiments. More results are underway which can contribute to understanding of basic phenomena in weakly ionized gases, combustion, and shock dynamics on the interfaces.

[1] A. Markhotok, S. Popovic, L. Vuskovic, Phys. of Plasmas J. **15**, 3 (2008).

L1.00025 On the origin of contact angle hysteresis , YUMIKO YOSHITAKI, Science and Engineering, Tokyo Denki University, KO OKUMURA, Department of Physics, Ochanomizu University — In this study, we consider a simple distribution of defects on a substrate, or a sinusoidal 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. 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, which is in contrast with the conventional theory where the CAH originates from the viscous dissipation inside the liquid around the contact line just after depinning.

L1.00026 Particle collision in Newtonian and viscoelastic fluids , AREZOO ARDEKANI, MIT — Particle-particle and particle-wall collision occurs in many natural and industrial applications such as sedimentation, crystal growth, suspension rheology, and microfluidic devices such as those used in mechanical cell lysis. To accurately predict the behavior of particulate flows, fundamental knowledge of the mechanisms of single collision is required. In this work, particle-wall collision in Newtonian and viscoelastic fluids is numerically and experimentally studied. The effect of Stokes number, surface roughness, and Deborah number on the rebound velocity of a colliding spherical particle on a wall is considered. The experimental study of particle-wall collision in poly(ethylene-oxide) mixed with water shows that the results for the coefficient of restitution in polymeric liquids can be collapsed together with the Newtonian fluid behavior if one defines the Stokes number based on the local strain rate. In addition, the effects of particle interaction and collision on the droplet breakup in a particulate shear flow are studied. The presence of particles leads to larger droplet deformation and a perforation in the center of the droplet. It is found that the critical Stokes number above which a perforation occurs increases linearly with the inverse of the capillary number and viscosity ratio.

L1.00027 Lagrangian Panel Method for Vortex Sheet Motion in 3D Flow[1] , ROBERT KRASNY, University of Michigan, HUALONG FENG, Nanjing University of Science and Technology, LEON KAGANOVSKIY, New College of Florida — A Lagrangian panel method is presented for computing vortex sheet motion in 3D flow. The sheet is represented by a set of quadrilateral panels with a quadtree structure. The panels have active particles that carry circulation and passive particles used for adaptive panel refinement. The Biot-Savart kernel is regularized and the particle velocity is computed using a treecode. Results are presented for the azimuthal instability of a vortex ring and the oblique collision of two vortex rings.

[1]Supported by NSF.

L1.00028 Mixing enhancement in a non-parallel microfluidic chip[1] , FANG YANG, CUIFANG KUANG, WEI ZHAO, GUIREN WANG, NANO/MICRO FLUIDICS LAB, DEPT. ME UNIVERSITY OF SOUTH CAROLINA TEAM — Electrokinetic instability (EKI) flow can be use as an efficient tool for mixing in a lab-on-a chip system. In this report, we fabricated a quasi T channels with electrodes on the sidewall to enhance mixing with AC electrokinetics. A parametric study was conducted to explore the effectiveness of manipulating EKI waves to enhance fluid mixing inside the microchannel channel. Firstly, mixing results in two cases have been compared: electrodes are placed at the sidewall and electrodes are located at the ends of the channel. Secondly, the mixing results in the microchannel with different angle between two electrodes were assessed in terms of scalar concentration distributions. Thirdly, the effectiveness of the applied voltage phase variation between the two electrodes on the mixing process inside the quasi T channel were also explored for the further mixing enhancement. Fourthly, mixing result under high frequency was also achieved. Finally fluorescent particles in one of the two streams were used to obtain a more clear visualization of mixing process in the microchannel with 5° angle between the two sidewalls.

[1]This work has been financially supported by the NSF RII funding (EPS-0447660).

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LA Turbulent Boundary Layers: Theory 101A

3:35PM LA.00001 Optimal linear amplification of partial energy norms in turbulent channels[1] , JAVIER JIMENEZ, U. Poltecnica de Madrid and CTR — When comparing "optimally-amplified" linearized porturbations in modeled turbulent channels to the observed structures in real multiscale flows, it is often useful to redefine the norm used in the comparison. For example, to study the kinetic energy at a given wall distance, it is useful to examine which initial perturbations amplify optimally the energy at that level, rather than the total one. The same is true of individual velocity components. Unfortunately, such norms are singular, vanishing for some nonzero perturbations, and require modifying the standard optimum-growth algorithm, We present such a modification, using a nonsingular norm to normalize the initial condition, and a possibly singular one to monitor growth, and apply it to model the energy spectra in a linearized eddy-viscosity approximation to turbulent channels. Using a norm that measures the buffer-layer energy leads to maximum amplification in the spectral region characteristic of near-wall streaks, while using energy windows farther from the wall isolates spectra similar to those in the log or outer layers. The results are used to investigate the likely origin of the different turbulent structures.

[1]Funded by CICYT.

3:48PM LA.00002 Optimal energy amplification of plane turbulent channel flows with emphasis on different types of perturbation , YONGYUN HWANG, ASHLEY WILLIS, CARLO COSSU, Ecole Polytechnique — Optimal perturbations in turbulent channel flow with mean velocity profile and its associated eddy viscosity are investigated, with emphasis on different types of perturbation. We look for linear amplification of both very large-scale outer structures and near-wall streaks arising from three types of perturbations: initial perturbation, harmonic forcing and stochastic excitation. Proper premultiplied energy amplification factors for optimal harmonic forcing and stochastic excitation are suggested to be identified with growth of the outer and inner structures. Finally, response of the turbulent flow to optimal perturbations is studied, based on direct numerical simulation to validate the linear analysis.

4:01PM LA.00003 Initial conditions and symmetry breaking for linear energy amplification in eddy-viscosity models of turbulent channels[1] , PABLO GARCÍA-RAMOS, U. Politécnica Madrid, JAVIER JIMÉNEZ, UPM and CTR Stanford — We study maximally-growing linear perturbations in a turbulent channel, using linearized Navier Stokes equations and an eddy viscosity that is generally lower than the one required to maintain the full velocity profile. The new viscosity depends on the wavenumber, and can be rationalized from spectral considerations. Significantly, fully nonlinear simulations using it, are able to self-sustain. We find that it is important to consider not only the optimal perturbation for a given wavenumber, but also those associated to the next few singular values. In general, these come in pairs. For short-wavelength modes localized near the wall, or in the sublayer, symmetric and antisymmetric eigenfunctions have essentially the same growth properties, showing that the two walls are decoupled. For larger wavelengths, whose optimal perturbations span the whole flow thickness, symmetry is broken, and the solution with an antisymmetric streamwise velocity becomes dominant. It corresponds to a fast streak in one wall opposite to a slow one in the other, and agrees with the structure of global modes obtained from correlations in full-channel DNSes.

[1]Funded by CICYT and the UPM.

4:14PM LA.00004 Log law via first principles[1] , JOE KLEWICKI, University of New Hampshire, PAUL FIFE, University of Utah, TIE WEI, LANL — The first-principles based theory of Fife et al. 2005 *J. Fluid Mech.* **532**, Fife et al. 2009 *J. Discrete & Cont. Dyn. Sys.* **24** is tested relative to the properties of the logarithmic mean velocity profile. The theory demonstrates that the mean momentum balance (MMB) formally admits a hierarchy of scaling layers, with an associated length scale distribution that asymptotically scales with distance from the wall. DNS data are shown to support these and other analytical findings. The mean velocity profile exhibits logarithmic dependence (exact or approximate) when the solution to the MMB exhibits (exact or approximate) self-similarity on the hierarchy. Exact self-similarity corresponds to a constant leading coefficient in the logarithmic mean velocity equation. An independent equation for this coefficient (von Kármán coefficient, κ), and its various equivalent forms are shown support by DNS data. Physically, κ exists owing to approximately scale invariant dynamics over an internal layer hierarchy. The theory clarifies how and why logarithmic dependence occurs and that logarithmic dependence is inherently approximate.

[1]This work was jointly supported by the NSF and ONR.

4:27PM LA.00005 On the nature of Karman coefficient variantion in wall-bounded turbulent flow , ZHEN-SU SHE, XI CHEN, YANZHI WANG, YOU WU, Peking University, China, SHE TEAM — It becomes increasingly recognized that the so-called Karman constant in wall-bounded flow is not universal, but varies from channel to pipe, and hence depends on the type of the boundary layers. Recent studies show that Karman coefficient vary with pressure gradients and unsteadiness of the smooth wall. In this study, we use an ensemble decomposition technique from a so- called structural ensemble dynamics theory, to analyze turbulent fluctuation data in a smooth channel. It is shown that a proper decompostion is able to characterize the mean property associated a set of turbulent structures. In particular, Karman coefficents obtained for bursts/streaks show systematic variations, which underlies the mechanism of variations of Karman coefficient with Reynolds numbers and with the geometry of the boundary layers.

4:40PM LA.00006 Recurrent dynamics in turbulent boundary layers[1] , DIVAKAR VISWANATH, University of Michigan — Recurrences, in which structures break-up, re-form and advect, are an important aspect of the fine scale motions in boundary layers. These recurrences can be captured using certain exact solutions of the Navier-Stokes equation, which are periodic in time but may or may not advect in space. We will use these solutions to demonstrate that spanwise advection is essential for the bursting phenomenon recorded in hotwire measurements near the wall. The importance of spanwise variation in the velocity field is well known. The new element in our work has to do with spanwise advection, not spanwise variation.

[1]NSF grants DMS-0715510

4:53PM LA.00007 A critical layer analogy for the very large scale motions in wall turbulence , BEVERLEY MCKEON, California Institute of Technology, ATI SHARMA, Imperial College London — A concatenation of experimental results in wall turbulence has shown the importance of the very large scale motions (VLSMs) in the streamwise direction, of the order of ten outer lengthscales. While Sreenivasan (1988) proposed a critical layer description of the variation of the wall-normal location of the peak in Reynolds shear stress, namely $y^+_{pk} \sim R^{+1/2}$, where $y^+ = yu_\tau/\nu$, in this presentation we extend the critical layer interpretation to explain the existence and scaling of the VLSMs. An analytical expression for the location of peak streamwise VLSM energy agrees well with experimental results, including the hot-wire results of Morrison et al (2004) from the Princeton/ONR Superpipe.

5:06PM LA.00008 The fluctuating topography and acceleration statistics in a turbulent channel flow , JOHN CHIRISTOS VASSILICOS, LU CHEN, Imperial College London, ZHIWEI HU, University of Southampton — Dallas, Vassilicos & Hewitt (PRE, 2009) characterised the turbulent channel flow topography of the fluctuating velocity field in terms of its stagnation points and quantified this topography by observing that, in the intermediate layer, the number density of stagnation points is inversely proportional to wall-distance. Our DNS of turbulent channel flow confirm this observation at skin friction Reynolds numbers 360 and 720. This spatial structure of the fluctuating velocity's topography partly determines the mean flow profile. We then study, in the intermediate layer, the motion of stagnation points which depends on the acceleration field. The mean streamwise acceleration equals the square of the skin friction velocity divided by the half width, the mean spanwise acceleration is zero and the mean wall-normal acceleration equals the vertical gradient of the mean square vertical fluctuating velocity. The local and convective acceleration terms tend to cancel each other and their rms values are equal only if the terms involving products of mean and fluctuating velocity terms are taken into account. When these terms are excluded, the rms of all three acceleration components are inversely proportional to wall-distance. Consequently,the rms of the streamwise stagnation point velocity is inversely proportional to wall-distance but the rms of this velocity's two other components are independent of it.

5:19PM LA.00009 Streamwise Constant Dynamics in Plane Couette Flow[1] , DENNICE GAYME, BEVERLEY J. MCKEON, California Institute of Technology, BASSAM BAMIEH, University of California, Santa Barbara, ANTONIS PAPACHRISTODOULOU, University of Oxford, JOHN C. DOYLE, California Institute of Technology — We have previously shown that when forced by small-amplitude Gaussian noise, a streamwise constant projection of the Navier Stokes equations captures many of the salient features of fully developed turbulent plane Couette flow. In this work we develop further the relationship between the nonlinearity in the model and the mathematical mechanism that results in the characteristic shape of the turbulent velocity profile. We use periodic spanwise-wall normal stream functions to represent an idealized model of the streamwise streaks and vortices that are thought to play an important role in both transition and fully developed turbulence in wall bounded shear flows. We demonstrate that using this model, such stream functions produce mean flows consistent with both DNS and experimental observations. Analysis of the amplification properties of the model around flow solutions arising from such stream functions is also studied in an effort to develop a quantitative bound on their energy contribution.

[1]Sponsored by a grant from the Boeing Corporation and NSF-CAREER Award number 0747672.

5:32PM LA.00010 Integral form of the skin friction coefficient suitable for experimental data[1] , CHRISTOPHER WHITE, FARAZ MEHDI, University of New Hampshire — An integral method to evaluate skin friction coefficient for turbulent boundary layer flow is presented. The method replaces streamwise gradients with total stress gradients in the wall-normal direction and is therefore useful in cases when velocity profiles at multiple locations are not available or feasible. It is also shown to be especially useful for experimental data with typical noisy shear stress profiles such as rough-wall boundary layer flows. This is significant, particularly in view of the limited ways by which skin friction can be determined for rough-wall flows.

[1]FM would like to acknowledge the support of the Office of Naval Research (N000140810836, grant monitor Ronald Joslin).

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LB Chaos and Fractals I 101B

3:35PM LB.00001 Driven Lid Flow in a Hemisphere: Steady-State Stokes Flow and 3-Dimensional Chaotic Advection Under Small and Large Perturbation , MICHAEL COFFEY, ANDREW OOI, University of Melbourne, DANIEL LESTER, GUY METCALFE, CSIRO Australia — We investigate flow and transport in a hemisphere filled with viscous fluid whose equitorial plane is a sliding lid. For steady-state Stokes flow the hemisphere has a single vortex line, nearly wall-attached, perpendicular to the sliding direction. Passive particle transport in the hemisphere is confined to cigar shaped shells that are continuously deformed from the vortex line to the boundary. We perturb the steady flow in two ways. (1) A small amplitude azimuthal oscillation in the sliding direction of the lid creates a fully three-dimensional dynamical system for particle transport in the hemisphere. Orbits between shells are connected through close approach manifolds at the parabolic points on the wall. (2) A periodically reoriented flow (PRF) occurs when the lid slides in one direction for a finite time τ then slides (without twisting) in a new direction reoriented from the first by a finite angle Θ. The PRF introduces symmetry, and we search for a mixing optimum in this stirred PRF.

3:48PM LB.00002 Moving Walls Accelerate Mixing[1] , JEAN-LUC THIFFEAULT, University of Wisconsin - Madison, EMMANUELLE GOUILLART, CNRS / Saint-Gobain Recherche, OLIVIER DAUCHOT, CEA Saclay — Mixing in viscous fluids is challenging, but chaotic advection in principle allows efficient mixing. In the best possible scenario, the decay rate of the concentration profile of a passive scalar should be exponential in time. In practice, several authors have found that the no-slip boundary condition at the walls of a vessel can slow down mixing considerably, turning an exponential decay into a power law. This slowdown affects the whole mixing region, and not just the vicinity of the wall. The reason is that when the ergodic mixing region extends to the wall, a separatrix connects to it. The approach to the wall along that separatrix is polynomial in time and dominates the long-time decay. However, if the walls are moving then closed orbits are created, separated from the bulk by a homoclinic orbit connected to a hyperbolic fixed point. The long-time approach to the fixed point is exponential, so we recover an overall exponential decay, albeit with a thin unmixed region near the wall.

[1]This work was partially supported by the Division of Mathematical Sciences of the US National Science Foundation, under grant DMS-0806821.

4:01PM LB.00003 The effect of finite Reynolds numbers on chaotic advection , PRADEEP RAO, ANDREW DUGGLEBY, Texas A&M University, PANKAJ KUMAR, MARK STREMLER, Virginia Tech — The effect of finite Reynolds numbers on chaotic advection is investigated for two dimensional lid-driven cavity flows that exhibit topological chaos in the creeping flow regime. The exponential convergence afforded by the use of spectral methods enables accurate tracking of passive scalars and exponential stretching of material lines in the flow. The emphasis in this endeavor is to study how the inertial effects present due to the low, but non-zero, Reynolds numbers produce irreversibilities in the system that affect the efficacy of mixing. Comparisons are made with Stokes flow results for the same configurations. The application of these results to systems that can effectively separate two substances with slightly different diffusivities in a medium are discussed.

4:14PM LB.00004 On the role of topological chaos and ghost rods in fluid mixing , MOHSEN GHEISARIEHA, MARK STREMLER, Virginia Tech — We consider stirring and mixing of two-dimensional Stokes flow in a circular domain due to the motion of three rods. Two similar protocols are discussed that are expected to give significantly different results based on the predictions of the Thurston-Nielsen (TN) theorem. Somewhat surprisingly, under many conditions the topologically "trivial" finite order protocol produces a larger stretch rate than does the pseudo-Anosov protocol, which is guaranteed to be chaotic by the TN theorem. We show that, in these cases, periodic points in the flow act as "ghost rods" that can be considered responsible for the large stretch rates produced by the finite order protocol. However, the existence and importance of these ghost rods is dependent on the specific system geometry, and perturbations can lead to very low stretch rates when using the finite order protocol. In contrast, selection of a pseudo-Anosov protocol leads to a robust minimum for the stretch rates as predicted by the TN theorem. In order to associate the stretch rate results to fluid mixing, we also discuss the homogenization of a passive scalar advected by the flow.

4:27PM LB.00005 Stirring with ghost rods in a lid-driven cavity , PANKAJ KUMAR, JIE CHEN, MARK STREMLER, Virginia Tech — It has shown that passive fluid particles moving on periodic orbits can be used to 'stir' a viscous fluid in a two-dimensional lid-driven cavity that exhibits a figure-eight flow pattern (Stremler & Chen 2007). Fluid motion in the vicinity of these particles produces "ghost rod" structures that behave like semi-permeable rods in the flow. Since these ghost rods are present due to the system dynamics, perturbations in the boundary conditions lead to variations in the existence and structure of the ghost rods. We discuss these variations and assess the role of ghost rods in mixing over a range of operating conditions for this system. The results suggest that ghost rods can play an important role in mixing for other counter-rotating flows.

4:40PM LB.00006 Almost-invariant sets as "ghost rods" for fluid stirring , MARK STREMLER, PANKAJ KUMAR, PIYUSH GROVER, SHANE ROSS, Virginia Tech — In two-dimensional time-dependent flows or three-dimensional flows with a certain symmetry, the braiding of periodic orbits provides a framework for analyzing chaos in the system through application of the Thurston-Nielsen (TN) classification theorem. "Ghost rods," or periodic orbits generated by the dynamics, behave as physical obstructions that "stir" the surrounding fluid, and these can be used as the basis for this topological analysis. We explore the identification of almost-invariant sets, or regions of fluid with high local residence time, as ghost rods. This set-oriented approach can be applied using relatively coarse system information, making this a promising approach for extending the use of the TN classification theorem to a variety of fluid systems.

4:53PM LB.00007 Fast computation of Lagrangian coherent structures: algorithms and error analysis , STEVEN BRUNTON, CLARENCE ROWLEY, Princeton University — This work investigates a number of efficient methods for computing finite time Lyapunov exponent (FTLE) fields in unsteady flows by approximating the particle flow map and eliminating redundant particle integrations in neighboring flow maps. Ridges of the FTLE fields are Lagrangian coherent structures (LCS) and provide an unsteady analogue of invariant manifolds from dynamical systems theory. The fast methods fall into two categories, unidirectional and bidirectional, depending on whether flow maps in one or both time directions are composed to form an approximate flow map. An error analysis is presented which shows that the unidirectional methods are accurate while the bidirectional methods have significant error which is aligned with the opposite time coherent structures. This relies on the fact that material from the positive time LCS attracts onto the negative time LCS near time-dependent saddle points.

5:06PM LB.00008 Topology counts: Statistics of critical points in experimental, two-dimensional flow[1] , DOUGLAS H. KELLEY, NICHOLAS T. OUELLETTE, Yale University — Points in a flowing fluid where the speed is zero — and therefore no streamline can be drawn — are known as critical points and have special topological significance. Two types exist in two-dimensional flows: hyperbolic (saddle) points and elliptic (center) points. Approximating two-dimensional flow with an electromagnetically driven, stably stratified solution in a 90 cm x 90 cm tray, we use particle tracking to measure the velocity field and locate the critical points. Our field of view encompasses $\sim$200 critical points per frame, each of which can be tracked like a particle over many frames. We will discuss the resulting spatiotemporal statistics of critical points in two-dimensional flow, focusing in particular on number fluctuations.

[1]This work is supported by the National Science Foundation.

5:19PM LB.00009 Chaotic mixing in vortex-dominated flows[1] , MARK KINGSBURY, TOM SOLOMON, Bucknell University — We present experimental studies of chaotic mixing in time-periodic, two-dimensional (2D) arrays of vortices. The flows are formed from the superposition of two vortex arrays shifted by half a vortex width in both directions. Flows generated by this method show both diffusive and superdiffusive transport, depending on the relative strengths and nature of the time dependence of the currents producing the two vortex arrays. Experimentally, we track the motion of tracer particles moving with the flow. From these tracks, we can determine the growth of the variance of a distribution of tracers. We are also applying algorithms based on braiding analysis [2] to determine the topological entropy for mixing in these flows. We are also initiating studies of chaotic mixing in a 3D, time-independent flow (composed of nested vortices) with the goal of studying advection-reaction-diffusion processes in a 3D system.

[1]Supported by NSF Grants DMR-0703635 and PHY-0552790.
[2]J.-L. Thiffeault, preprint

5:32PM LB.00010 Burning lobes in an advection-reaction-diffusion system[1] , DELORA GASKINS[2], TOM SOLOMON, Bucknell University — We use the tools developed for chaotic fluid transport to describe the propagation of a reaction front in a chain of oscillating vortices. Specifically, we expand the concept of lobes (turnstiles) bounded by stable and unstable manifolds to account for the propagation of a reaction front. A "burning lobe" is a passive lobe that grows due to the reactive medium. We propose that a reaction front will propagate from one vortex to the next in an oscillation period if a portion of the front was within the burning lobe. This framework is used to explain mode-locking of reaction fronts propagating in an oscillating vortex array.

[1]Supported by NSF Grants DMR-0703635 and PHY-0552790.
[2]Current Address: California State University, Long Beach

Monday, November 23, 2009 3:35PM - 5:32PM –
Session LC Turbulence Modelling II 101C

3:35PM LC.00001 Evaluation of Turbulence Closures from a DNS Database of Shock Boundary Layer Interaction , 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 first solve the mean flow equations using the turbulence statistics obtained from a DNS database (Wu & Martin, 2008); we alter the Reynolds stresses by perturbing the turbulent time scale and the stress anisotropy to investigate the effect of potential modeling errors on the resulting skin friction and wall pressure. As a second step, we use the same methodology to study the effect of modeling assumptions in the turbulent transport equations. We also discuss the use of the invariant map to introduce Reynolds stress perturbations that remain realizable.

3:48PM LC.00002 A priori comparison of RANS scalar flux models using DNS data of a Mach 5 boundary layer , KALEN BRAMAN, VENKATRAMANAN RAMAN, Department of Aerospace Engineering and Engineering Mechanics, The University of Texas at Austin — In order to investigate the applicability of Reynolds-averaged scalar flux models (SFM) to scalar dispersion in high speed turbulent flows, a priori comparisons have been performed utilizing the results of direct numerical simulations (DNS) of a Mach 5 boundary layer. At a small patch on the solid surface boundary, a scalar was introduced into the flow at a rate depending upon the local surface temperature. This configuration mimics surface ablation in hypersonic flows. In different simulations, the scalar injection rate was varied, and the scalar was treated as both passive, not affecting the flow field, and active, affecting the flow field due to having different molecular properties than the bulk flow and having an injection velocity. Statistics of the simulated scalar fields have been calculated and compared a priori with terms from SFMs. Comparisons from the passive scalar case show that the scalar flux terms in the standard gradient diffusion model fail to predict even the trend of the DNS values. The generalized gradient diffusion models, while an improvement for the streamwise component of scalar flux, nevertheless fail to predict the wall normal and spanwise fluxes. Additionally, production and dissipation models for the scalar variance equation are evaluated.

4:01PM LC.00003 The Influence of Spanwise Rotation on the Redistribution of Turbulent Kinetic Energy in Fully-Developed Channel Flows , CHARLES PETTY, KARUNA KOPPULA, ANDRE BENARD, Michigan State University — A recently developed universal, realizable, anisotropic prestress (URAPS-) closure for the normalized Reynolds (NR-) stress is used to predict the influence of spanwise rotation on the components of the NR-stress in fully-developed channel flows. Direct numerical simulation (DNS-) results are used to determine the relative time scales needed to solve the non-linear URAPS-equation. The new closure, which predicts the existence of a region of zero intrinsic vorticity on the high pressure side of the channel, provides an explanation of how rotation redistributes turbulent kinetic energy among the three components of the fluctuating velocity. The non-linear algebraic URAPS-equation is formulated as a mapping of the NR-stress into itself; therefore, fixed points of the URAPS-equation are *realizable* for all turbulent flows, regardless of the benchmark flows used to calibrate the model parameters. The URAPS-closure together with closed transport equations for the turbulent kinetic energy and the turbulent dissipation provide a low-order closure for the RANS-equation.

4:14PM LC.00004 Uncertainty Quantification for RANS Turbulence Model Predictions[1] , TODD OLIVER, ROBERT MOSER, The University of Texas — We examine the application of a Bayesian uncertainty quantification (UQ) framework to Reynolds-averaged Navier-Stokes (RANS) turbulence model predictions. The UQ framework involves the solution of a statistical inverse problem, where probability density functions (pdfs) for the parameters of a chosen stochastic model are calibrated, and a statistical forward problem, where the uncertainty represented by the calibrated pdfs is propagated through the model to the quantity of interest (QoI). In the context of RANS models, which are deterministic, a significant challenge is the development of an appropriate stochastic extension of the deterministic physical model. This stochastic extension is required to enable the model structural uncertainty—i.e., the uncertainty in the form of the physical model—to be represented in the inverse problem and properly propagated in the forward problem. Multiple possible formulations of a stochastic extension of popular RANS models will be discussed, and preliminary results in modeling turbulent boundary layers will be shown.

[1]This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

4:27PM LC.00005 Optimal harmonic forcing and drag reduction in pipe flow , ASHLEY P. WILLIS, YONGYUN HWANG, CARLO COSSU, LadHyX, Ecole Polytechnique — A simple (RANS) model is used in combination with and explicit expression for the eddy viscosity, to model the linear response of turbulent pipe flow to harmonic forcing. Although the response is not so large as for the laminar case, it is shown that the greatest response still occurs for forcing on the largest scale, i.e. on the scale of the size of the domain. As this scale is independent of the flow rate, any strategy for drag reduction on this scale should be practical over a large range of Reynolds numbers. Also, as any drag reduction technique must modify the flow, it is convenient that it the large-scale motion is the most efficiently induced. The only question remaining, therefore, is whether the large scale motion can lead to drag reduction in this geometry. In this work we show that forcing large scale motions can produce competitive drag reduction and at little energetic cost.

4:40PM LC.00006 A New Turbulence Model for Capturing Physics in Variable-Density Flows , KRISTA STALSBERG-ZARLING, ROBERT GORE, RICK RAUENZAHN, Los Alamos National Laboratory — Variable-density flows are ubiquitous in nature. They are encountered on a larger scale in astrophysical, geophysical, atmospheric and oceanic flows, as well as on a smaller level, for example, in inertial confinement fusion and engine combustion. Our goal was to develop a turbulence model that is universally applicable across a wide variety of variable-density configurations by including more of the physics important to these types of flows. As part of this effort, the BHR turbulence model, which we use to close the traditional RANS equations, has been extended to capture the effects of molecular mixing in miscible fluids, as well as changes in turbulence quantities that occur due to de-mixing. Here, we demonstrate these new model capabilities for a flow that evolves from an unstable to stable Rayleigh-Taylor state. The model results are validated against experimental data.

4:53PM LC.00007 Effects of streamline curvature on separation prediction , SUNIL K. AROLLA, PAUL A. DURBIN, Department of Aerospace Engineering, Iowa State University — In this study, the effects of streamline curvature on prediction of flow separation are investigated. The geometry is a circulation control airfoil, a high-lift configuration that has been under extensive research for more than two decades. A tangential jet is blown over a thick, rounded trailing edge, using the Coanda effect to delay separation. An attempt is made to understand, through numerical simulations, the dynamics of turbulent separation and reattachment on the Coanda surface. Highly curved, attached recirculation regions are seen to form. A physics based curvature correction proposed by Pettersson-Reif et al. (1999) is used in conjunction with $\zeta - f$ turbulence model. The chord-based Reynolds number is $Re = 10^6$. Two jet momentum coefficients of $C_\mu = 0.03$ and 0.1 are computed. In this paper, comparisons between the computed and experimental pressure distributions, velocity profiles and the position of flow detachment are presented. Comparisons with other closures such as Menter's SST model are also discussed.

5:06PM LC.00008 Turbulent Flow Past Spinning Cylinders[1] , IGBAL MEHMEDAGIC, DONALD CARLUCCI, PASQUALE CARLUCCI, U. S. Army ARDEC, SIVA THANGAM, Stevens Institute of Technology — Flow past cylinders aligned along their axis where a base freely spins while attached to a non-spinning forebody is considered from a computational and experimental point of view. The time-averaged equations of motion and energy are solved using the modeled form of transport equations for the turbulence kinetic energy and the scalar form of turbulence dissipation with an efficient finite-volume algorithm. An anisotropic two-equation Reynolds-stress model that incorporates the effect of rotation-modified energy spectrum and swirl is used to perform computations for the flow past axially rotating cylinders. Both rigid cylinders as well as that of cylinders with free-spinning base are considered from a computational point of view. A subsonic wind tunnel with a forward-sting mounted spinning cylinder is used for experiments. Experiments are performed for a range of spin rates and free stream flow conditions. The experimental results of Carlucci & Thangam (2001) are used to benchmark flow over spinning cylinders. The data is extended to munitions spinning in the wake of other munitions. Applications involving the design of projectiles are discussed.

[1]This work was funded in part by U. S. Army ARDEC.

5:19PM LC.00009 Predicting Law-of-the-Wall with LES: Role of SFS and Surface Stress Models[1] , JAMES BRASSEUR, TIE WEI, SANJIV RAMACHANDRAN, Penn State Univ. — In previous work we presented a framework in which large-eddy simulation (LES) can be designed to capture the law-of-the-wall (LOTW) in mean velocity gradient when the first grid level is in the inertial layer. A proper combination of grid aspect ratio (A_R), subfilter-scale (SFS) stress model constant (C_t), and vertical grid resolution is required to suppress a spurious frictional length scale that underlies deviations from LOTW scaling. This occurs when the LES exceeds critical values of: (1) ratio of mean resolved to SFS shear stress (R) at the first grid level, (2) an "LES Reynolds number" (Re_{LES}), and (3) vertical resolution—the "High Accuracy Zone" (HAZ) on a plot of R vs. Re_{LES}. Here we demonstrate this framework for 2 eddy viscosity and one non eddy viscosity SFS models, and we show that for the eddy viscosity models both R and Re_{LES} are inversely proportional to $D_t = C_t^a A_R^b$, where a and b are model-dependent constants. Commonly applied surface shear stress models create a spurious sink in velocity variance and oscillations in mean velocity gradient at the surface. Correcting the spurious sink both reduces these oscillations and increases the predicted value of the Von Kármán constant (VKC). We also show that C_t and A_R must be chosen within certain bounds and that the VKC predicted by LES asymptotes to a value of about 0.37 within these bounds when the LES is within the HAZ.

[1]Supported by ARO.

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LD Flow Control VI 101D

3:35PM LD.00001 Assessment of Closed-Loop Control Using Multi-Mode Sensor Fusion For a High Reynolds Number Transonic Jet[1] , KERWIN LOW, BASMAN ELHADIDI, MARK GLAUSER, Syracuse University — Understanding the different noise production mechanisms caused by the free shear flows in a turbulent jet flow provides insight to improve "intelligent" feedback mechanisms to control the noise. Towards this effort, a control scheme is based on feedback of azimuthal pressure measurements in the near field of the jet at two streamwise locations. Previous studies suggested that noise reduction can be achieved by azimuthal actuators perturbing the shear layer at the jet lip. The closed-loop actuation will be based on a low-dimensional Fourier representation of the hydrodynamic pressure measurements. Preliminary results show that control authority and reduction in the overall sound pressure level was possible. These results provide motivation to move forward with the overall vision of developing innovative multi-mode sensing methods to improve state estimation and derive dynamical systems. It is envisioned that estimating velocity-field and dynamic pressure information from various locations both local and in the far-field regions, sensor fusion techniques can be utilized to ascertain greater overall control authority.

[1]Air Force Office of Scientific Research

3:48PM LD.00002 An Adjoint Solver for Unsteady Navier-Stokes Flows and Application in Uncertainty Quantification , QIQI WANG, Massachusetts Institute of Technology, FRANK HAM, GIANLUCA IACCARINO, PARVIZ MOIN, Stanford University — We present a parallel adjoint solver for unsteady incompressible Navier-Stokes equation. Backward time-stepping for the time-dependent adjoint equation is achieved using the dynamic checkpointing scheme. This adjoint solver is suitable for very long time integration without knowing the number of time steps a priori. The required computational time and memory is only three to five time the solution of the forward Navier-Stokes equation for tens of thousands of time steps. The adjoint solver is tested on a laminar cylinder vortex shedding calculation at Reynolds number 100. Application of the adjoint solver in propagating uncertainties with large number of random variables is also presented.

4:01PM LD.00003 Direct Numerical Simulation of Forced Round Jets , MURALIDHAR KRISHNAMURTHY, Professor, Department of Mechanical Engineering, Indian Institute of Technology Kanpur 208016, TRUSHAR GOHIL, Doctoral student, ARUN SAHA, Assistant Professor — Jet control leading to noise reduction and efficient combustion can be achieved by manipulating its coherent structures. In the present study, direct numerical simulation of free circular (round) jets has been performed for two types of forcing, namely, flapping (FLP) and dual-mode excitation (DME). A Reynolds number of 1000 based on average jet velocity and nozzle diameter is considered. A small scale perturbation of 5% rms velocity is initially added to the three velocity components. DME perturbation is obtained by combining an axisymmetric excitation at the preferred mode frequency and helical excitation at a frequency fixed by the disturbance frequency ratio. For both contexts, the amplitude of the large scale excitation is 15% of the base velocity. The finite difference representations are second order accurate in time, fourth order in the advection term and sixth order in diffusion. With small-scale perturbation, well-defined vortex rings are shed at the preferred mode frequency. With DME, a spectacular increase of jet spreading is seen on one of the orthogonal planes. On the other hand, for the FLP perturbation, the circular jet shows a bifurcation pattern that is practically a Ψ-shape.

4:14PM LD.00004 The interaction of an array of circumferentially varying stators with a uniform crossflow[1] , JOHN FARNSWORTH, MICHAEL AMITAY, Rensselaer Polytechnic Institute — A propuslor capable of producing maneuvering forces in all directions effectively eliminates the need for additional control surfaces. Side forces can be generated by the propeller through the variation of the inflow swirl velocity to a conventional propeller. These control forces are generated based on the same geometric principles as a helicopter swash-plate. Instead of cyclically adjusting the propeller blade pitch angle, the relative pitch angle of a fixed pitch propeller is cyclically altered through a preswirled inflow generated by an upstream stator row. Wind tunnel experiments were conducted where global and detailed flow measurements were acquired through surface static pressure, and stereoscopic PIV on a simplified propulsor model. From these measurements a better understanding of the fluidic interactions associated with the non-uniform upstream stator row and the flow field was achieved.

[1]Supported by the ONR University Lab Initiative Program.

4:27PM LD.00005 Controlled Transitory Flow Attachment over a Stalled Airfoil[1] , GEORGE WOO, ARI GLEZER, Georgia Institute of Technology — Controlled attachment of transitory stall over a pitching airfoil is investigated in wind tunnel experiments using an array of surface-integrated pulsed jet actuators. The actuation has a characteristic time scale that is an order of magnitude shorter than the convective time scale of the base flow and results in momentary flow attachment with significant temporal changes in circulation and consequently in the aerodynamic forces and moments. The flow field in the cross stream plane above the pitching airfoil and in its near wake is investigated using high-resolution PIV phase-locked to the actuation. A single actuation pulse results in transitory flow attachment that is manifested by rapid increase in the global circulation and aerodynamic forces and persists for about ten convective time scales before the flow becomes fully stalled again. Large-scale changes in vorticity accumulation that are associated with repetitive, burst-modulated actuation pulses are exploited for significant extension of the streamwise domain and duration of the attached flow that is coupled with an increase in the peak circulation. Measurements of the interaction between the pulsed jets and the cross flow reveal details of the severing and collapse of the separated flow domain, and the dynamics of vorticity accumulation within the attaching boundary layer.

[1]Supported by NASA's Subsonic Rotary Wing Program.

4:40PM LD.00006 3-D Interactions of Synthetic Jets and Cross-Flows - Experiments[1] , JOSHUA WOOD, MICHAEL AMITAY, Rensselaer Polytechnic Institute — The interaction of synthetic jets with a cross-flow over a finite wing was studied experimentally at a low Reynolds number and low angles of attack using PIV. The focus of the work was to explore the details of the flow structures near the synthetic jets. The interaction of the synthetic jets with the flow resulted in an array of counter-rotating vortical structures. The pair of counter-rotating vortices formed at the jet orifice was found to be two-dimensional just downstream of the jet orifice; however, as it advected downstream, it developed three-dimensionalities. The effect of the momentum coefficient (or blowing ratio) on the 3-D interaction was also explored; at low momentum coefficient only a slight interaction between the vortices was observed; however, as the momentum coefficient increases, the vortices interacted with each other, causing them to lift off the surface. Furthermore, the spanwise extent of the coherent structures was reduced as they advected downstream.

[1]Supported by AFOSR grant number FA9550-08-1-0233, monitored by Dr. John Schmisseur.

4:53PM LD.00007 The Lift Response of a Stalled Wing to Pulsatile Disturbances[1] , DAVID WILLIAMS, Illinois Institute of Technology, GILEAD TADMOR, Northeastern University, TIM COLONIUS, California Institute of Technology, WESLEY KERSTENS, VIEN QUACH, SETH BUNTAIN, Illinois Institute of Technology — The transient lift response of a low Reynolds number wing subjected to small amplitude pulsatile disturbances is investigated. The wing has a semi-circular planform, and is fully stalled at a 20° angle of attack. Micro-valve actuators distributed along the leading edge of the wing produce the transient disturbance. It is shown that the lift response to a single pulse increases with the square root of the increasing actuator supply pressure, and that the lift response curves are similar to each other when scaled by the total impulse. Furthermore, for fixed actuator supply pressure, the amplitude and total impulse of the transient lift response curve increases with increasing external flow speed. In this case the lift response curves are similar when scaled by the dynamic pressure. The lift response to a single pulse can be treated as a filter kernel, and can be used to predict the lift time history for arbitrary actuator input signals. Comparisons with multiple-pulse inputs and continuous actuation modulated at low frequencies show good agreement between the model predictions and the experiment.

[1]Support by AFOSR grants FA9550-05-1-0369 and FA9550-09-1-0189 is gratefully acknowledged.

5:06PM LD.00008 A Moving Airfoil Controlled by Synthetic Jets[1] , SOL KEUN JEE, OMAR LOPEZ, ROBERT MOSER, University of Texas at Austin, ALI KUTAY, JONATHAN MUSE, ANTHONY CALISE, Georgia Institute of Technology — There is a growing interest in synthetic jets in flow controls. Here we consider the use of synthetic jet actuators in active control of an airfoil. An adaptive controller is integrated with a CFD model which includes details of the synthetic jet actuators and detached eddy simulation for turbulent flows at Re=900,000. This integration allows us to investigate a moving airfoil controlled by synthetic jets. Two synthetic jet actuators are mounted on the top and bottom of a NACA 4415 airfoil close to the trailing edge to generate bi-directional aerodynamic moment. Aerodynamic performance is explored using the closed-loop controller to regulate 2-degree-of-freedom motions of the airfoil. The coupled CFD/controller model simulates maneuverings of the airfoil as studied experimentally in wind tunnel tests. Modeling of unsteady aerodynamics with synthetic jets is validated against measurements in the wind tunnel including aerodynamic forces, surface pressure distributions and PIV velocity fields. The coupled model also demonstrates rapid maneuvers on the order of the convecting time scale. The characteristics of the unsteady aerodynamics coupled with the vehicle dynamics and the synthetic jets will be discussed.

[1]This work is supported by AFOSR grant FA9550-05-1-0411.

5:19PM LD.00009 Coanda-assisted Spray Manipulation , KATIE MABEY, BARTON SMITH, REID ARCHIBALD, BRIAN WEST, Utah State University — An overview of research on a flow control technique called Coanda-assisted Spray Manipulation (CSM) is presented. CSM uses a high-momentum control jet under the influence of the Coanda effect to vector a high volume-flow jet or spray. Actuators provide the capability of moving the location of applied control flow making rotary or arbitrary motion of the vectored flow possible. The presented work includes a fundamental isothermal study on the effects of rotation speed and Reynolds number on a vectored jet using a belt-driven CSM actuator. Three-component velocity data were acquired for three Reynolds numbers and three rotation speeds using timed resolved high-speed stereo Particle Image Velocimetry. A second CSM system with 16 pneumatically-driven control ports has been retrofitted to a flame spray gun. This combination provides the capability to rapidly alter the direction of applied metal powders. High speed video of this process will also be presented. Finally, a fundamental study on the pneumatic system's response to minor losses and connection lines of varying lengths is presented.

5:32PM LD.00010 Active Vibration Control of an S809 Wind Turbine Blade Using Synthetic Jet Actuators[1] , VICTOR MALDONADO, MATTHEW BOUCHER, REBECCA OSTMAN, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Active flow control via synthetic jet actuators was implemented to improve the aeroelastic performance of a small scale S809 airfoil wind turbine blade model in a wind tunnel. Blade vibration performance was explored for a range of steady post-stall angles of attack, as well as various unsteady pitching motions for a chord based Reynolds number range of 1.29×10^5 to 3.69×10^5. Blade tip deflection was measured using a pair of calibrated strain gauges mounted at the root of the model. Using flow control, significant vibration reduction was observed for some steady post-stall angles of attack, while for dynamic pitching motions, vibration reduction was more pronounced (for a given angle of attack) on the pitch up motion compared to the pitch down motion of the blade cycle. This effect was attributed to the phenomenon known as dynamic stall, where the shedding of a leading edge vortex during the pitch up motion contributes to elevated values of lift (compared to static angles of attack) and lower values of lift when the blade is pitched down. This effect was also quantified through the use of Particle Image Velocimetry.

[1]Funding provided by the New York State Energy Research and Development Authority (NYSERDA).

Monday, November 23, 2009 3:35PM - 5:32PM –

Session LE Biofluids VII: Phonation/Glottal Flows 101E

3:35PM LE.00001 Flow Induced Vibration and Glottal Aerodynamics in a Three-Dimensional Laryngeal Model[1] , XUDONG ZHENG, QIAN XUE, RAJAT MITTAL, Johns Hopkins University, STEVEN BIELAMOWICZ, George Washington University — Three-dimensional effects associated with phonation remain unclear due to the lack of capability of simulating 3D fluid-tissue interaction in the past. To advance the state-of-the-art in this arena, an immersed-boundary method based flow solver coupled with a finite-element solid dynamics solver is employed to conduct high-fidelity direct-numerical simulations of phonation in a 3D model of the human larynx. Three-dimensional vibration patterns are captured along with turbulence effects and three-dimensional vortex structures in the glottal jet. Results from these simulations are presented.

[1]Supported by NIDCD Grant R01 DC007125-01A1

3:48PM LE.00002 Computational Modeling and Analysis of Phonation in a Diseased Larynx[1] , QIAN XUE, XUDONG ZHENG, RAJAT MITTAL, Johns Hopkins University, STEVEN BIELAMOWICZ, George Washington University — The goal of our current research is to study the fundamental biophysics of phonation in healthy and pathological larynges. We have developed a coupled flow-structure interaction solver to help gain insight into the fundamental biophysics underlying unilateral laryngeal paralysis. The solver models the incompressible Navier-Stokes equations for the glottal aerodynamics and employs the classic two-mass model for the vocal folds. The effect of tension imbalance and subglottal pressure on the vocal fold dynamics is investigated. An analysis of the vibration modes as well as the frequency spectra and sound quality has been conducted. Results from these studies will be presented.

[1]Supported by NIDCD Grant R01 DC007125-01A1.

4:01PM LE.00003 Simulation of Vocal Folds: A Fluid-Induced Self-Oscillating Problem , XINGSHI WANG, LUCY ZHANG, Department of Mechanical, Aerospace & Nuclear Engineering Rensselaer Polytechnic Institute — The goal of this study is to investigate the process of voice production by simulating the motion and deformation of human vocal folds. The vocal folds are oscillated by a constant lung pressure driven airflow in the throat. The system is modeled in 2-D using the immersed finite element method to simulate and study the fluid-structure interaction mechanism. From our numerical results, the glottal jets are identified. Several parameters such as the Reynolds number, Strouhal number, vocal folds stiffness, density ratio between the fluid and the structure are addressed and compared with experimental results. The frequency of the vocal folds vibration, fluid flow rate and pressure distribution are also investigated. In addition, the energy transfer between the fluid domain and the solid domain are analyzed to assist in explaining the underlying physical mechanism for this fluid-induced self-oscillating vocal folds.

4:14PM LE.00004 The development of supraglottal flow structures during speech[1] , BYRON ERATH, Purdue University, MICHAEL PLESNIAK, George Washington University — During voiced speech, periodic vocal fold oscillations create a pulsatile jet that emanates from the glottis and is convected through the supraglottal tract. Unsteadiness in the supraglottal jet trajectory (superior to the vocal folds) has been observed in a variety of laryngeal flow investigations, contributing to sound production due to vortex pairing which occurs within the jet as well as the impingement of the deflected jet on physiological structures. However, there is confusion in the literature concerning the mechanisms which contribute to the flow variability. Instabilities in the supraglottal jet include the Kelvin-Helmholtz instability, as well as the hypothesized presence of jet 'flip-flopping', the tendency of the glottal jet to detach from one vocal fold wall and reattach to the opposing wall mid-cycle. The morphology of the supraglottal flow field is investigated using phase-averaged PIV measurements acquired in the anterior-posterior midplane of the superior vocal fold tract of a dynamically controlled 7.5 time life-size vocal fold model. Flow parameters are scaled to match physiological values. The relevant fluid flow phenomena that impact the supraglottal jet trajectory are identified.

[1]Supported by the National Science Foundation under Grant No. CBET-0828903.

4:27PM LE.00005 Three-dimensional flow patterns in a scaled, physical vocal fold model with a unilateral polyp[1] , ANGELA SEAWRIGHT, BYRON ERATH, Purdue University, MICHAEL PLESNIAK, George Washington University — Trauma to the vocal folds often causes the formation of polyps; affecting the efficiency of speech and making voice rough and breathy. The change in flow characteristics due to a unilateral polyp positioned on the medial surface of a 7.5 times life-size physical vocal fold model was investigated. Previously reported phase-averaged intraglottal particle image velocimetry (PIV) investigations in a coronal plane indicated significant variations in the flow behavior on different anterior offset planes relative to the polyp. Flow three-dimensionality was investigated by resolving the temporal evolution of the flow with laser Doppler velocimetry (LDV). Data were acquired superior to the glottal exit. Physiological values of Reynolds, Strouhal, and Euler numbers were matched. Results were compared to velocity fields generated by healthy vocal fold motion. The glottal jet trajectory, flow separation points, and the velocity distribution along the vocal fold walls were influenced. Thus, a polyp significantly disturbs and modifies the airflow through the vocal folds, which has implications on both the fluid-structure energy exchange and the sound production.

[1]Supported by the National Science Foundation under Grant No. CBET-0828903.

4:40PM LE.00006 Measurement of Glottal Flow across Scaled Up Dynamic Vocal Fold Motion , ERICA SHERMAN, RPI, MICHAEL KRANE, Penn State ARL, LUCY ZHANG, TIMOTHY WEI, RPI — An experiment to provide DPIV measurements of dynamic human vocal folds motion is presented. The experiment is run in a free-stream water tunnel using a 10x scaled-up model of the human vocal folds and vocal tract. The vocal fold model is a new design that incorporates both the rocking as well as the oscillatory open/close motions characteristic of vocal fold motions The Reynolds number and Strouhal number have been matched to human physiologic conditions. Flow measurements show the start-up jet, vortex dynamics and ultimate jet pinch-off as the model progresses through a cycle. The effects of asymmetries associated with disease will be discussed.

4:53PM LE.00007 Aeroelastic-aeroacoustic measurements in a self-oscillating physical model of the human vocal folds[1] , MICHAEL KRANE, ARL Penn State, ZACHARY CATES, ARL Penn State/Virginia Tech — Measurements are presented characterizing the relationship between the structure of physical models of the human vocal folds and the sound produced by their vibration by airflow from the lungs. The model vocal folds are fabricated by molding two layers of silicone rubber of specified stiffness, approximating the body/cover structure. These are mounted in a model vocal tract, where the prephonatory gap adjusted using micropositioners. Measurements conducted in an anechoic chamber include radiated sound pressure, and high-speed video of the vibrating model vocal folds, using prephonatory separation, body stiffness, and subglottal pressure as input parameters.. Essential behavior of the vocal fold models is presented. Vibration fundamental frequency and radiated sound pressure level outside the model vocal tract as a function of subglottal pressure and prephonatory gap are presented for the cases of two identical vocal folds and one vocal fold with lower stiffness, approximating vocal fold paralysis.

[1]Acknowledge support of NIH and ARL Penn State.

5:06PM LE.00008 Direct measurement of aeroacoustic source spectrum due to a jet/wall interaction[1] , SHANE LANI, MICHAEL KRANE, ARL Penn State — The aeroacoustic source spectrum due to a turbulent jet passing over an obstruction is found experimentally. The model consists of a constriction and planar obstacle in a duct with dimensions commensurate with the those of a human vocal tract. An unsteady jet formed at a constriction interacts with a planar obstruction downstream with the jet normal to the planar surface. The aeroacoustic source spectrum is found both by measuring the unsteady force imparted on the planar obstruction as well as measuring radiated sound outside the duct. A comparison of the force spectrum to the inverse-filtered radiated sound measurements will be presented.

[1]Acknowledge support of NIH and PSU-ARL.

5:19PM LE.00009 Comparison of measured aeroacoustic source spectra to predictions using a jet model[1] , DANIEL LEONARD, MICHAEL KRANE, ARL Penn State — Sound radiated from a turbulent jet-wall interaction in a duct is measured for several jet-wall interaction geometries, for which the acoustic response of the duct was identical at low frequencies. This sound production mechanism is identical to that of unvoiced speech sounds. Traditionally in these cases, the speech science community has stressed the acoustic filter's role in determining the radiated sound, and has neglected the importance of the aeroacoustic source. When the local source region aerodynamics, such as the mean jet path relative to the wall and the jet speed are varied, but the acoustic filter held constant, distinct differences due to the aeroacoustic source are observable in the radiated sound. The source spectra are determined and qualitatively compared to an analytical model, and the distinct differences in the source spectra are described theoretically, whereas the classical approach would not have been able to theoretically describe these results. It is concluded that the turbulent jet's path makes a crucial contribution to the 'shape' of the source spectrum and that unvoiced speech sound production depends as much on the local details of the source region aerodynamics and geometry as it does on the acoustic filter.

[1]Acknowledge support from NIH and PSU-ARL E&F program.

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LF Microfluidics: Drops and Capillarity 101F

3:35PM LF.00001 Lubrication failure of viscous threads in microfluidic chambers[1] , SAMIRA DARVISHI, THOMAS CUBAUD, Stony Brook University — We investigate lubricated transport of high-viscosity fluids in microfluidic systems. Using miscible liquids having disparate viscosities, we produce a viscous core transported within a less viscous annulus, i.e., a viscous thread, in a square microchannel. Downstream, the thread motion is studied in diverging-converging slit microchannels. We describe a variety of flow patterns resulting from the folding instability and examine the relationships between flow morphologies and system parameters including fluid viscosities, mass diffusion coefficient, flow rates, and micro-cell geometry. In particular, we demonstrate that small threads can traverse the extension without lubrication failure while large threads experience a significant dilation due to direct contact with the top and bottom walls. We also investigate the lubrication failure of capillary threads using immiscible fluids and show the possibility to manipulate contact lines.

[1]This work is supported by NSF (CBET-0932925).

3:48PM LF.00002 Folding of capillary threads in microfluidic networks[1] , THOMAS CUBAUD, SAMIRA DARVISHI, Stony Brook University — We examine the evolution of the folding instability of lubricated viscous threads in straight microchannels. Folds having a uniform wavelength can be produced using a diverging microchannel connected to three channels in parallel. This design allows for the detailed experimental study of the influence of viscosity contrast, interfacial properties, and flow rates on the structure of miscible and immiscible micro-threads. In particular, we focus on the spatial damping of the amplitude of sinuous capillary threads due to interfacial tension effects. This study shows methods for the interfacial control of high-viscosity fluids in microfluidic systems.

[1]This work is supported by NSF (CBET-0932925).

4:01PM LF.00003 Nanoemulsion Through Stretching-Folding Instability , CLAUS-DIETER OHL, CHON U. CHAN, Nayang Technological University — A new kind of instability sets in when a oil filament is focused by surrounding water-flow through a thin constriction commonly used in fluid focusing devices. At sufficiently high flow rates the oil filament is forced into harmonic oscillations through interfacial forces. Past the constriction the liquid is suddenly slowed down which leads to rapid shortening of the filament's wavelength. At sufficiently high amplitudes the co-flowing water stream breaks up and pinches off micrometer and sub-micrometer sized droplets in a very repeatable manner, thus producing a water-in-oil emulsion. This pinch off is caused by a stretching and folding instability when the oscillating flow impinges into the quasi stagnant reservoir past the constriction. We will present high-speed movies at up to 300,000 frames per second resolving details of the very fast events. A simple model based on restoring interfacial forces is able to predict the kilohertz oscillation frequency observed.

4:14PM LF.00004 Droplet formation and storage using immiscible two liquids in a micro-channel with transverse micro-ribs , JIHOON KIM, DOYOUNG BYUN, YOUNGHUN GWAK, Konkuk University, JONGIN HONG, Imperial College London — Such a transition in the micro-channel with micro-ribs was also studied, which may be significant in designing super-hydrophobic micro-channel. An in-depth study of the wetting transition in micro-channel with micro-ribs is carried out to scrutinize the condition of the wetting transition. And based on the optimized condition for the wetting transition, we investigate the flow characteristics of two immiscible liquids in the micro-channel in order to generate and storage the droplet. When the interface of immiscible liquids moves across the cavity between two neighboring micro-ribs, the oil phase may replace the water in the cavity, isolating the water phase in a corner of the cavity and forming a droplet. The isolated water volume directly affects the droplet size, which is determined by the speed of the interface, the geometry of the micro-ribs, and physical properties of the fluids, such as viscosity and surface tension. For the formation of uniform droplets without any daughter droplets, the synchronization of both contacts on the top of the forward micro-rib and the bottom of the cavity must be considered to find for optimal condition.

4:27PM LF.00005 Generating double emulsions W/O/W in PDMS systems for pharmaceutical applications , HERVE WILLAIME, Laboratoire MMN UMR7083 CNRS-ESPCI, NADIA SNOUSSI, Laboratoire de Physique Pharmaceutique UMR CNRS 8612-Université Paris-Sud — Vesicular systems, and especially water/oil/water multiple emulsions, present fascinating properties of pharmaceutical interest (release of fragile pharmaceutically active molecule, detoxification). Their implementation requires a strict control of their governing physical parameters, since their properties (stability, efficiency) crucially depends on the size of the internal droplets, and of the globules containing these droplets. However, traditional methods for generating emulsions with the help of high shear mixers do not allow to produce calibrated systems, thus limiting their characterization. It is well known that microfluidics provides a prominent tool to generate simple or multiple emulsions in a controlled way, with a very low diameter dispersion (below 5%). In this talk, we will present preliminary results concerning the generation of such objects (sizes distributions, stabilities...) and compare them to traditional methods.

4:40PM LF.00006 Scaling the drop size in coflow experiments , ELENA CASTRO-HERNANDEZ, Area de mecanica de fluidos, Universidad de Sevilla, VENKATA GUNDABALA, ALBERTO FERNÁNDEZ-NIEVES, School of Physics, Georgia Institute of Technology, JOSE MANUEL GORDILLO, Area de mecanica de fluidos, Universidad de Sevilla, AREA DE MECANICA DE FLUIDOS, UNIVERSIDAD DE SEVILLA TEAM, SCHOOL OF PHYSICS, GEORGIA INSTITUTE OF TECHNOLOGY TEAM — We performed extensive experiments with coflowing fluids in microfluidic devices. When the inner fluid is a liquid, two different types of regimes have been identified, dripping and jetting. Dripping is characterized by the fact that no long jets of the dispersed phase are formed. By contrast, when jetting occurs, the dispersed phase forms long liquid jets and drops are emitted right at the tip of the liquid thread. In the jetting regime, we could reproduce the *widening* and *stretching* regimes [Utada et. al. PRL 99 (2007)]. We have given a step further and provide a general expression to estimate the drop size in either regime as a function of measurable parameters. By contrast to the liquid case, when the inner liquid is a gas, we find that no long jets form, irrespective of the values of the control parameters, [Marin et. al. Coll. Surf. A (2009)].The crucial role of the axial strain exerted by the outer stream on the inner one to stabilize long fluid threads will be elucidated by means of BEM simulations, which show good agreement with experiments.

4:53PM LF.00007 Droplet break-up in microfluidic T-junctions at small capillary numbers , MARIE-CAROLINE JULLIEN, ESPCI-CNRS, MARY JANE TSANG MUI CHING, CÉLINE COHEN, LAURE MÉNÉTRIER, ESPCI, PATRICK TABELING, ESPCI-CNRS, MICROFLUIDICS, MEMS AND NANOSTRUCTURES TEAM — We perform experimental studies of droplet breakup in microfluidic T-junctions in a range of Capillary numbers lying between 4.10^{-4} and $2\ 10^{-1}$ and for two viscosity ratios of the fluids forming the dispersed and continuous phases. The present paper extends the range of Capillary numbers explored by previous investigators by two orders of magnitude. We single out two different regimes of breakup. In a first regime, a gap exists between the droplet and the wall before breakup occurs. In this case, the break up process agrees well with the analytical theory of Leshansky and Pismen [Phys. Fluids, 21(2), 023303 (2009)]. In a second regime, droplets keep obstructing the T-junction before breakup. Using physical arguments, we introduce a critical droplet extension for describing the breakup process in this case.

5:06PM LF.00008 Hydrodynamics and Heat Transfer of Discrete Droplets in Microfluidic Devices , ROBERT WEBER, SHERVIN SHAJIEE, KAMRAN MOHSENI, University of Colorado at Boulder — Electrostatic manipulation of surfaces tension forces is now a standard fluid handling technique in microfluidic devices. In this investigation electrowetting on dielectric (EWOD) is employed in order to use discrete droplets for thermal management of compact micro systems. Both hydro- and thermodynamics of digitized droplets are investigated by experimental, theoretical and computational means. EWOD devices have been built on silicon substrates with highly doped layers replacing metal electrodes, and higher quality thermal oxides replacing the more expensive PECVD oxides. In parallel, an experimental test rig has been built to measure the heat transfer rate of the slug flow at a macro scale. Droplets at several length and speed are created systematically. Average heat transfer rates and Nusselt numbers in constant heat flux in a tube has been experimentally measured for continuous and discrete water flow cases and the results have been compared with numerical results.

5:19PM LF.00009 A numerical method for Stokes flow with capillary effects , JOHN FETTIG, JONATHAN FREUND, LUKE OLSON, University of Illinois — A simulation tool is presented for capillary driven reacting and polymerizing low-Reynolds-number flows. The free surface is represented by a level-set function, but with extra terms added so that it remains sharp despite finite numerical diffusion. The governing equations are discretized with hierarchical finite elements. A splitting facilitates implicit time advancement of the nonlinear advection-diffusion transport equation. At small Capillary numbers, the pressure jump at the free surface due to surface tension makes the saddle-point discrete system for the velocity expensive to solve. A decomposition into two parts, facilitated by the hierarchical elements, significantly accelerates the overall solution. The first part governs the static system, which includes the sharp interface. This portion requires no incompressibility constraint and is therefore relatively easily solved using p refinement for high accuracy. The second part is dynamic but smooth, so it can be solved in relatively few iterations despite the incompressibility constraint. This decomposition reduces the computational expense, by up to 95% in the demonstration simulations, which are relevant to self-healing autonomic materials.

5:32PM LF.00010 Novel Method for Measuring Temperature of Microchannel Flows Using Polydiacetylene Sensor Droplets , IMSUNG YOO, SIMONG SONG, Hanyang Univ. — Monitoring temperature in a microchannel flow is important when to use a microfluidic chip for biochemical analysis like cell culture. There are two typical methods: using thin film thermocouples (TFTCs) and Rhodamine B solutions. The former can measure temperature of a microchannel flow accurately, but requires complex fabrication processes and high costs. In addition, it disturbs microchannel flows and is contaminated readily by reaction between TFTC and flow solutions. On the other hand, Rhodamine B has the excellent sensor property that its fluorescent intensity is linear to a flow temperature. Unfortunately, however, it often adsorbs to microchannel surfaces like PDMS and has issues of bio-compatibility due to its hydrophobicity Thus, we propose a novel method to monitor temperature of a microchannel flow using polydiacetylene (PDA) sensor droplets. PDA, a conjugated polymer, has a unique property to transform its color from visible blue to fluorescent red by thermal stress. By monitoring the fluorescence intensity of PDA droplets in a microchannel, we found a linear relationship between the flow temperature and the fluorescence intensity in a certain temperature range.

Monday, November 23, 2009 3:35PM - 5:45PM –

Session LG Nano-Fluids I 101G

3:35PM LG.00001 Three-phase contact line at small scale , ANTONIO PEREIRA, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, UK — We investigate the area around an equilibrium three-phase contact line at a small scale by using a density functional approach. A typical system is made of a planar wall in contact with a Lennard-Jones gas below the critical temperature. The wall exerts an attractive force on the fluid molecules so that a thin film can usually form between the wall and the gas. We focus on two cases. When the chemical potential is smaller than its coexistence value and the system presents a phase transition with respect to the film thickness, we examine the area between the two equilibrium film thicknesses. It appears to be smooth and several molecular diameters long. When the chemical potential is at its coexistence value, computations of the equilibrium density profiles show a well formed contact angle whose value follows closely the Young equation.

3:48PM LG.00002 ABSTRACT WITHDRAWN –

4:01PM LG.00003 Mechanical role of phospolipid bilayers in synovial joint lubrication[1] , ROSS PACKARD, YVES DUBIEF, LEONIE COWLEY, School of Engineering, University of Vermont, Burlington VT — Cartilaginous surfaces in synovial joints have been shown to be coated by multilamellar water and phospholipid bilayer arrangements. The interactions between the bilayers and synovial gel composed of hyaluronic acid (HA) and albumin forms the basis of the SAPL (surface active phospholipids) theory to explain joint lubrication. While considerable efforts have been devoted to the rheological properties of synovial gel, the ability of SAPL to sustain significant mechanical stress has received little attention. Using Gromacs molecular dynamics (MD) solver, we have characterized the response to compression and shear of multilamellar arrangements composed of water and dipalmitoylphosphatidylcholine (DPPC, a phospholipid in highest concentration in synovial fluid). MD simulations were performed with and without HA to isolate the dynamics that allow the multilamellar arrangement to retain its anisotropic structure beyond the normal gait pressure of 1.5 MPa. The critical normal compression that provokes the rupture of membranes and drainage of the system is a strong function of the thickness of the water layer and may exceed 15MPa due to complex underlying membrane dynamics.

[1]This work is supported by VT Epscor and Vermont Advanced Computing Center (supported by NASA-NNX-08AO96G).

4:14PM LG.00004 Understanding the structural properties of clusters in sheared aggregating systems using Brownian dynamics simulation , SERGIY MARKUTSYA, RODNEY FOX, DENNIS VIGIL, SHANKAR SUBRAMANIAM, Iowa State University — Nanoparticle synthesis in turbulent reactors subjects anoparticle aggregates to a homogeneous, time-varying shear flow. The shear flow results in anisotropic clusters and it is of interest to characterize the structural properties of these clusters and their effects on initiation and acceleration of aggregation, the restructuring of clusters, and their breakage. The anisotropic structure of a sheared cluster is characterized by the ratio of the major to minor axis length of the approximating ellipsoid oriented along the cluster moment of inertia tensor's principal axes. Brownian dynamics simulations show that shear flow dramatically changes the structure of aggregates by initiating the formation of more compact structures at smaller length scales perpendicular to the shear direction, and anisotropic, cigar–like structures along the shear direction. More compact clusters correspond to higher local volumetric potential energy density. Therefore, we classify the compactness and anisotropy of sheared clusters on a map of local volumetric potential energy density versus ratio of the principal values of the cluster's moment of inertia tensor. The effect of shear on breakage of clusters is characterized by the radius of gyration R_g^{cr} of the largest stable aggregate for a given value of the imposed steady shear rate (Péclet number).

4:27PM LG.00005 Nanodroplet impact onto solid platinum surface: Spreading and bouncing , DANIEL LUSSIER, YIANNIS VENTIKOS, Dept. Engineering Science, Univ. of Oxford — The impact of droplets onto solid surfaces is found in a huge variety of natural and technological applications, from rain drops splashing on the pavement, to material manufacturing by molten droplet deposition. Taking inspiration from existing microfluidic technologies (i.e. lab-on-chip), there is increasing interest in the use of nanodroplets (D $<$ 100 nm) for a number of applications such as drug delivery and semiconductor device manufacturing. However, as the size of the droplet is reduced into the nanoscale, the direct use of previously obtained macroscopic results is not guaranteed. At the nanoscale, important effects due to the molecular nature of the fluid, thermal fluctuations and reduced dimensionality can play a critical role in determining system dynamics. In this paper we present the results of large-scale, fully atomistic, three-dimensional molecular dynamics (MD) simulation of an argon nanodroplet (D = 18 nm, 54 000 atoms) impact onto a solid platinum surface, using the LAMMPS software package. The fluid argon is modeled using the well-known Lennard-Jones (LJ) potential, while the embedded-atom model (EAM) potential is used for the solid platinum. By varying both the impact velocities (10-1000 m/s) and the wettability of the solid surface a wide range of impact behaviors is observed, from smooth spreading, to bouncing recoil, pointing towards a wide array of potential applications.

4:40PM LG.00006 Properties of Nanofluids[1] , GIANLUCA PULITI, SAMUEL PAOLUCCI, MIHIR SEN, University of Notre Dame — Equilibrium molecular dynamic simulations are presented for different configurations of interactions between gold metal and liquid water. It makes use of state-of-the-art potentials to capture a broad spectrum of realistic physical phenomena at the interface. Thermodynamic properties, such as internal energy, heat capacities, isothermal compressibility, and coefficient of thermal expansion of the nanofluid are currently being analyzed. Transport properties, such as mass diffusion, viscosity and thermal conductivity, are also under investigation. Some of the results obtained thus far, seems to strongly diverge from the prediction of ideal mixture theories. The understanding of basic thermodynamic and transport effects in nanofluids is a stepping stone to further studies.

[1]Funding for this research is provided by DOE grant DE-FG36-08GO88020.

4:53PM LG.00007 Viscous Heating in Nanoscale Shear Driven Flows , BOHUNG KIM, ALI BESKOK, Old Dominion University — Three-dimensional Molecular Dynamics (MD) simulations of heat and momentum transport in liquid Argon filled shear-driven nano-channels are performed using 6-12 Lennard-Jones potential interactions. Work done by the viscous stresses heats the fluid, which is dissipated through the channel walls, maintained at isothermal conditions via a recently developed interactive thermal wall model. Momentum transport in shear driven nano-flow is investigated as a function of the surface wettability ($\varepsilon_{wf}/\varepsilon$), spatial variations in the fluid density, kinematic viscosity, shear- and energy dissipation rates are presented. Temperature profiles in the nano-channel are obtained as a function of the surface wettability, shear rate and the intermolecular stiffness of wall molecules. The energy dissipation rate is almost a constant for $\varepsilon_{wf}/\varepsilon < 0.6$, which results in parabolic temperature profiles in the domain with temperature jumps due to the well known Kapitza resistance at the liquid/solid interfaces. Using the energy dissipation rates predicted by MD simulations and the continuum energy equation subjected to the temperature jump boundary conditions developed in [Kim et al., *Journal of Chemical Physics*, 129, 174701, 2008], we obtain analytical solutions for the temperature profiles, which agree well with the MD results.

5:06PM LG.00008 Understanding the nanoscale liquid-solid interfacial phenomena in a Couette flow , XIN YONG, LUCY ZHANG, Rensselaer Polytechnic Institute — Molecular dynamics simulations are used to study a nanoscale Couette flow and investigate the slip behavior at liquid-solid interfaces. We model liquid argon confined between two smooth rigid copper walls and the upper wall velocity is imposed to shear the fluid slab. The velocity fields, density distributions, flow boundary conditions and liquid structures/orderings are studied for walls ranging from hydrophobic to hydrophilic. We observe various flow boundary conditions ranging from pure slip to multi-layer locking in the simulations. The results show that, temperature, liquid-solid interaction parameter and shear rate are the major factor influencing the fluid structures at liquid-solid interfaces, thus determine the flow boundary condition. Different liquid states are named which are characterized by different liquid structures. Continuous transitions between states are also found. Furthermore, the positive correlation between shear rate and slip length is established and the temperature shift of the shear rate-slip length curve is observed. We confirm the unbounded slip length at high shear rates and correlates it to momentum transfer mechanism between liquid and solid atoms. In transient simulations, liquid may transit from several metastable states to a ground state under certain conditions.

5:19PM LG.00009 Electrokinetic transport in realistic nanochannels[1] , MORAN WANG, EES16 and CNLS, Los Alamos National Laboratory, JIN LIU, Department of Bioengineering, University of Pennsylvania, QINJUN KANG, EES-16, Los Alamos National Laboratory — When an electrolyte solution contacts with a solid surface, the surface will likely be charged through an electrochemical adsorption process. The surface charge in general varies with the local bulk ionic concentration, the pH value and the temperature of the solution, and even with the double layer interactions in the narrow channel. Most of the previous studies are based on a constant zeta potential or surface charge density assumption, which does not reflect the realistic charge status at interfaces and may lead to inaccurate predictions. In this work, we first develop a generalized model for electrochemical boundary conditions on solid-liquid interfaces, which can closely approximate the known experimental properties. We further present nonequilibrium molecular dynamic (NEMD) simulations of electrokinetic transport in nanochannels. We take silica and carbon as examples of channel materials. Both monovalent and multivalent ionic solutions are considered. The electrokinetic transport properties for realistic nanochannels are therefore studied and a multiscale analysis for a new energy conversion device is performed.

[1]This work is supported by LANL's LDRD Project 20080727PRD2.

5:32PM LG.00010 Molecular Dynamics Simulation of Shock Waves Interacting with Nanostructures , AHMAD ALQANANWAH, JOEL KOPLIK, YIANNIS ANDREOPOULOS — Typical theoretical treatments of shock wave interactions are based on a continuum approach, which cannot resolve the spatial variations in solids with nano-scale porous structure. To investigate such interactions we have developed a molecular dynamics simulation model, based on Lennard-Jones interactions. A piston, modeled as a uni-directional repulsive force field translating at a prescribed velocity, impinges on a region of gas which is compressed to form a shock, which in turn is driven against an atomistic solid wall. Periodic boundary conditions are used in the directions orthogonal to the piston motion, and we have considered solids based on either atoms tethered to lattice sites by stiff springs, or on embedded atom potentials. Velocity, temperature and stress fields are computed locally in both gas and solid regions, and displacements within the solid are interpreted in terms of its elastic constants. In this talk we present preliminary results, and the longer-term goal of this work is to understand energy transport and absorption in porous materials.

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LH Drops VI 101H

3:35PM LH.00001 Conditions for Destabilizing Pickering emulsions using external electric fields , KYUHO HWANG, PUSHPENDRA SINGH, NADINE AUBRY — Fine particles are readily adsorbed at fluid-fluid interfaces, and can be used as stabilizers in emulsion technology by preventing adjacent drops from coalescing with each other. We investigate a new technique to destabilize such emulsions, or Pickering emulsions, by applying an external electric field. Experiments show that the latter has two effects: (i) the drops elongate in the direction of the electric field, (ii) the local particle density varies on the drop surface due to the dielectrophoretic (DEP) force acting on the particles. It is shown that the latter is the dominant factor in the destabilization process. Particularly, the success of the method depends on the values of certain dimensionless parameters; specifically, the ratio of the work done by the dielectrophoretic force must be larger than the work done by the buoyant force. Moreover, drops do not coalesce through the regions where the particles locally cluster, whether those are gathered at the poles or at the equator of the drops. As particles move, particle-free openings form on the drop's surface, which allow for adjacent drops to merge. This process takes place even if the particles are fully packed on the drops' surfaces as particles get ejected from the clustering areas due to a buckling phenomenon.

3:48PM LH.00002 Deformation and Rotation of a Drop in a Uniform Electric Field[1] , PAUL SALIPANTE, JAMES HANNA, PETIA VLAHOVSKA, Thayer School of Engineering, Dartmouth College — Drop deformation in uniform electric fields is a classic problem. The pioneering work of G.I.Taylor demonstrated that for weakly conducting media, the drop fluid undergoes a toroidal flow and the drop adopts a prolate or oblate spheroidal shape, the flow and shape being axisymmetrically aligned with the applied field. However, recent studies have revealed a nonaxisymmetric rotational mode for drops of lower conductivity than the surrounding medium, similar to the rotation of solid dielectric particles observed by Quincke in the 19th century. We will present an experimental and theoretical study of this phenomenon in DC fields. The critical electric field, drop inclination angle, and rate of rotation are measured. For small, high viscosity drops, the threshold field strength is well approximated by the Quincke rotation criterion. Reducing the viscosity ratio shifts the onset for rotation to stronger fields. The drop inclination angle increases with field strength. The rotation rate is approximately given by the inverse Maxwell-Wagner polarization time. We also observe a hysteresis in the tilt angle for low-viscosity drops. The effects of AC fields and surfactants are also explored.

[1]Supported in part by ACS-PRF grant 48415-G9.

4:01PM LH.00003 Dynamics of rotating high viscous droplet by using electrostatic levitator , TOMOYUKI MAEKAWA, Tsukuba University, SATOSHI MATSUMOTO, Japan Aerospace Exploration Agency, YUTAKA ABE, AKIKO KANEKO, Tsukuba University, TADASHI WATANABE, Japan Atomic Energy Agency, KATSUHIRO NISHINARI, University of Tokyo — The electrostatic levitation is one of the containerless processing techniques. Thermo-physical properties of extreme high temperature molten metals have been measured with an oscillating drop method based on linear approximations. For example, a viscosity has been estimated from the damping constant of oscillation after the deformation imposed. However, this method is limited to the low viscosity fluid because the oscillation is not excited on viscous liquid drop. Thus, the objective is to develop a new method to measure the wide range of viscosity liquid. In the present study, several different droplets with different viscosity were levitated by the electrostatic force. Drop deformation was imposed applying rotation. Dynamics of rotating droplet was investigated. The time dependence of drop midpoint radius for drop breakup when the rotation speed increased was experimentally measured. The effect of viscosity on the deformation was made clear.

4:14PM LH.00004 Co-flowing liquids in the presence of electric fields , ALBERTO FERNANDEZ-NIEVES, VENKAT GUNDABALA, Georgia Tech, IGNACIO GONZALEZ-LOSCERTALES, University of Malaga, ANTONIO BARRERO, University of Sevilla — We apply electric fields to a liquid that is extruded through a capillary tip in the presence of a surrounding co-flowing liquid and induce formation of a jet that ultimately breaks into drops. We will present some results related to this experiment, which can lead to a better and extended control of the drop sizes.

4:27PM LH.00005 Analysis of Electrowetting Dynamics with Level Set Method , JUN KWON PARK, JIWOO HONG, KWAN HYOUNG KANG, Department of Mechanical Engineering, Pohang University of Science and Technology, San 31, Hyoja-dong, Pohang 790-784 — Electrowetting is a versatile tool to handle tiny droplets and forms a backbone of digital microfluidics. Numerical analysis is necessary to fully understand the dynamics of electrowetting, especially in designing electrowetting-based liquid lenses and reflective displays. We developed a numerical method to analyze the general contact-line problems, incorporating dynamic contact angle models. The method was applied to the analysis of spreading process of a sessile droplet for step input voltages in electrowetting. The result was compared with experimental data and analytical result which is based on the spectral method. It is shown that contact line friction significantly affects the contact line motion and the oscillation amplitude. The pinning process of contact line was well represented by including the hysteresis effect in the contact angle models.

4:40PM LH.00006 Prediction of Time Response of Electrowetting[1] , SEUNG JUN LEE, JIWOO HONG, KWAN HYOUNG KANG, Pohang University of Science and Technology — It is very important to predict the time response of electrowetting-based devices, such as liquid lenses, reflective displays, and optical switches. We investigated the time response of electrowetting, based on an analytical and a numerical method, to find out characteristic scales and a scaling law for the switching time. For this, spreading process of a sessile droplet was analyzed based on the domain perturbation method. First, we considered the case of weakly viscous fluids. The analytical result for the spreading process was compared with experimental results, which showed very good agreement in overall time response. It was shown that the overall dynamics is governed by P2 shape mode. We derived characteristic scales combining the droplet volume, density, and surface tension. The overall dynamic process was scaled quite well by the scales. A scaling law was derived from the analytical solution and was verified experimentally. We also suggest a scaling law for highly viscous liquids, based on results of numerical analysis for the electrowetting-actuated spreading process.

[1]This work was supported by the Korea Research Foundation Grant funded by the Korean Government (MOEHRD, Basic Research Promotion Fund) (KRF-2006-331-D00058).

4:53PM LH.00007 Electro-hydrodynamic printing of drugs onto edible substrates[1] , YUEYANG SHEN, EZINWA ELELE, PRASHANTH PALLE, BORIS KHUSID, New Jersey Institute of Technology, OSMAN BASARAN, PATRICK T. MCGOUGH, ROBERT T. COLLINS, Purdue University — While most existing drugs are manufactured as tablets using powder processing techniques, there is growing interest in printing drops containing pharmaceutical actives on edible substrates. We have developed a drop-on-demand (DOD) printing method appropriate for either replacing existing manufacturing platforms or enabling personalized medicine that overcomes the various critical challenges facing current DOD technologies. To eliminate adverse effects of electro-chemical reactions at the fluid-electrode interface, the fluid is infused into an electrically insulating nozzle to form a pendant drop that serves as a floating electrode capacitively coupled to external electrodes. A liquid bridge is formed and broken as the voltage applied at the electrode is varied in time. This gentle method for drop deposition has been demonstrated to operate with fluids spanning over three orders of magnitude in viscosity and conductivity. The proposed method has the potential for the evolving field of pharmaceutical and biomedical applications requiring the deposition of fluids at the exact locations with high volume accuracy.

[1]The was supported by the NSF ERC for Structured Organic Particulate Systems.

5:06PM LH.00008 Theoretical and experimental study of meniscus behavior under AC electric field for Electrohydrodynamic (EHD) jetting , SI BUI QUANG TRAN, DOYOUNG BYUN, Konkuk University, Seoul, Korea — The electrohydrodynamic (EHD) spraying technique has been utilized in applications such as inkjet printing and mass spectrometry technologies. In this paper, the role of electrical potential signals in jetting and on the oscillation of the meniscus is evaluated. The jetting and meniscus oscillation behavior are experimentally investigated under ac voltage, ac voltage superimposed on dc voltage, and pulsed dc voltage. Furthermore, the analytical simulation about the oscillation of an anchored edge hemispherical meniscus located on a conductive flat plate under a uniform ac electric field is presented. The mutual interaction between the electric field and the hydrodynamics is iteratively solved. As a result, the simulation can calculate the meniscus shapes, contours of voltage outside the meniscus and the velocity profile of liquid inside the meniscus during the period of the oscillation according to the applied frequency. Based on the present theory, one can predict the oscillation mode with a certain applied frequency. The present theory can also be applied to investigate the oscillation of a free conductive drop in a uniform ac electric field.

5:19PM LH.00009 Electrohydodynamic ejection without using nozzle electrode , VU DAT NGUYEN, DOYOUNG BYUN, Konkuk University — The electrohydrodynamic (EHD) ejection technique has been applied to inkjet printing technology for fabrication of printed electronics. The conventional EHD inkjet device is based on dc voltage and requires two electrodes: a nozzle electrode and an extractor electrode. This study notes several drawbacks of the conventional EHD printing device such as electrical breakdown and demonstrates stable jetting by using the extractor electrode alone without the nozzle electrode and ac voltage. The continuous ejection of droplets can be obtained only by ac voltage, showing consistent ejection at every peak of electrical signal. The suggested EHD inkjet device prevents electrical breakdown and broaden the range of material selection for nozzle design. Experiments with high speed camera also point out that the generated droplets are much smaller than the nozzle size. Using glass capillary, we show various printing patterns of lines and characters.

5:32PM LH.00010 Large Deformation Studies of Vesicles under Electric Fields[1] , ROCHISH THAOKAR, PRIYA GAMBHIRE, IIT Bombay — Phospholipids tend to assume various liquid crystalline phases in water. One such phase, the liposome, forms an excellent model to study the properties of the (phospholipid) membranes. Liposomes have been subjected to electric fields and their deformation studied extensively, by various groups. Theydeform into prolate or oblate shapes based on the frequency of the applied field(alternating fields) or they undergo poration and fusion with the adjacent vesicles. We carry out both experimental and numerical studies on liposome deformation under applied AC electric fields. The small deformation regime agrees with the literature results. We study the deformation modes using high speed imaging. The behavior of liposomes under large field is complicated and dependent upon the properties of the fluids and the lipid membrane. The flow pattern within the medium in the liposome is investigated using flow markers. The large deformations are investigated using the Boundary integral method and comparisons made with the experimental observation. The phase lag between the applied AC field and the deformation response of the membrane is investigated and possibilities of the method as an interfacial rheometer discussed.

[1]Department of Science and Technology, India

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LJ Surface Tension Effects I 101I

3:35PM LJ.00001 Surfactant spreading on a thin fluid layer: visualization via fluorescence , DAVID FALLEST, North Carolina State University, CHRISTOPHER FOX, Harvey Mudd College, ELLEN PETERSON, MICHAEL SHEARER, KAREN DANIELS, North Carolina State University — We perform quantitative measurements of the spreading of an insoluble surfactant on a thin layer of glycerin. We directly observe both the radial height profile of the spreading droplet and the spatial distribution of the fluorescently-tagged surfactant during the spreading process. The spreading circular layer of surfactant forms a capillary ridge at the leading edge, the peak of which spreads with $R \approx t^{1/4}$, in agreement with predictions based on the lubrication approximation of the Stokes equations (Jensen & Grotberg, 1992). In addition, the surfactant concentration is observed to have a peak which follows, then lags behind, the capillary ridge during spreading. The companion talk (to follow) will compare these results with numerical simulations and analysis.

3:48PM LJ.00002 Surfactant spreading on a thin liquid layer: Modeling and Theory , ELLEN PETERSON, MICHAEL SHEARER, KAREN DANIELS, DAVID FALLEST, NCSU — To model the spreading of a droplet of insoluble surfactant on a thin liquid layer, we use the lubrication approximation of the Stokes equations. The resulting system of nonlinear PDE describes the height of the fluid surface and the distribution of surfactant on the surface. The flow is driven by the surface tension gradient induced by the surfactant. Ignoring the smoothing effects of capillarity and diffusion of surfactant, the system simplifies to a pair of transport equations. These equations admit a similarity solution characterized by Jensen and Grotberg (1992) that sets the spreading rate of the surfactant layer. We employ finite difference simulations to capture the shape of the free surface and the surfactant distribution. The leading edge of the surfactant layer is treated as a free boundary, with a boundary condition corresponding to a jump in the free surface height first discovered by Gaver and Grotberg (1990). We are able to examine a solution for both the outer region as well as the boundary layer that accounts for the behavior of the film near the center of the surfactant droplet. For the full system, with capillarity, gravity, and diffusion on the surface, the leading shock smooths to a ridge. We compare simulations of the full thin film system with experimental observations of film height profiles, and with the observed spreading rate of the surfactant layer.

4:01PM LJ.00003 Efficient numerical computation of fluid interfaces with soluble surfactant (I) an inviscid drop , MICHAEL BOOTY, MICHAEL SIEGEL, New Jersey Institute of Technology — We address a difficulty in the computation of fluid interfaces with soluble surfactant. At the large values of bulk Peclet number typical of fluid-surfactant systems, a transition layer forms adjacent to the interface in which the surfactant concentration varies rapidly, while its gradient at the interface must be evaluated accurately to determine bulk-interface exchange of surfactant, surface tension, and the drop's dynamics. We present a fast and accurate hybrid numerical method that incorporates a separate singular perturbation reduction of the transition layer into a full numerical solution of the interfacial free boundary problem. Results are presented for an inviscid drop in the Stokes flow limit, where the underlying flow solver for insoluble surfactant uses conformal mapping techniques in two dimensions. This facilitates comparison of the hybrid method with a traditional numerical method at moderately large Peclet number.

4:14PM LJ.00004 Efficient numerical computation of fluid interfaces with soluble surfactant (II) a viscous drop , KUAN XU, MICHAEL SIEGEL, MICHAEL BOOTY, New Jersey Institute of Technology — We address a difficulty in the computation of fluid interfaces with soluble surfactant. At the large values of bulk Peclet number typical of fluid-surfactant systems, a transition layer forms adjacent to the interface in which the surfactant concentration varies rapidly, while its gradient at the interface must be evaluated accurately to determine bulk-interface exchange of surfactant, surface tension, and the drop's dynamics. We present a fast and accurate hybrid numerical method that incorporates a separate singular perturbation reduction of the transition layer into a full numerical solution of the interfacial free boundary problem. Results are presented for a drop of arbitrary viscosity in the Stokes flow limit, where the underlying flow solver for insoluble surfactant uses a direct (primitive variable) boundary integral method.

4:27PM LJ.00005 Dynamic contact angles of evaporating liquids on heated surfaces , VLADIMIR AJAEV, Southern Methodist University, TATIANA GAMBARYAN-ROISMAN, TU Darsmtadt (Germany), JILL KLENTZMAN, University of Arizona, PATER STEPHAN, TU Darmstadt (Germany) — We studied dynamic apparent contact angle for gravity-driven flow of volatile liquid down a heated inclined plane. The apparent contact line is modeled as the transition region between the mascroscopic film and ultra-thin adsorbed film dominated by disjoining pressure effects. Three commonly used disjoining pressure models are investigated. The dynamic contact angle follows the Tanner's law remarkably well over a range of evaporation conditions. However, deviations from the predictions based on the Tanner's law are found when interface shape changes rapidly in response to rapid changes of the heater temperature. The Marangoni stresses are shown to result in increase of the values of apparent contact angles. Applications of different models of partial wetting to studies of fingering instability in evaporating liquids are also discussed.

4:40PM LJ.00006 The effect of evaporation and Marangoni stresses on contact line instability , JILL KLENTZMAN, University of Arizona, VLADIMIR AJAEV, Southern Methodist University — We investigate the contact line instability for the flow of viscous liquid on a heated inclined plane. Numerical simulations are conducted based on a model of the flow derived in our previous work using a lubrication-type approach including the effects of evaporation of the liquid. The impact of Marangoni stresses on the instability development is examined, and the value of the inclination angle of the solid surface is varied to study its effect. Unlike evaporation, which was shown in our previous work to inhibit the instability, Marangoni stresses promote the instability. Stability criteria for different inclination angles are expressed in terms of the modified Marangoni number and the evaporation number, a nondimensional measure of the mass flow rate across the interface. As a result of the development of instability, a finger-like structure is formed. The fingers grow initially, but then saturate at a length which depends on the evaporation conditions.

4:53PM LJ.00007 A Numerical Method for Variable Surface Tension Effects in Non-Isothermal Atomization[1] , PETER BRADY, JUAN LOPEZ, MARCUS HERRMANN, Arizona State University — Atomization often occurs in non-isothermal environments, such as in combustion devices. There, thermal fluctuations can be significant on length scales associated with the liquid atomization process. Since the surface tension force is a function of local temperature, these thermal fluctuations may result in large local variations of the surface tension force, thereby potentially impacting the atomization process. Here, we present a numerical technique to incorporate these thermal Marangoni forces into the balanced force Refined Level Set Grid (RLSG) approach. With it, the liquid/gas phase interface is tracked by a level set method using an auxiliary high resolution equidistant Cartesian grid. This not only allows for application of higher-order WENO schemes retaining their full order of accuracy both for advecting and reinitializing the level set scalar, but it also provides the necessary high resolution of the phase interface geometry during topology change events in an efficient manner. Verification and validation test cases geared towards testing the applicability of the proposed methods to the case of secondary spray drop atomization will be presented.

[1]This work is supported by NSF grant DMS-0808045.

5:06PM LJ.00008 The flow induced dynamic surface tension effects at nanoscale , ALEX LUKYANOV, University of Reading — The aim of this report is to describe in general the effects of dynamic surface tension solely induced by the flow over nanoscale topography of the substrates. Capillary effects of similar nature induced by chemical modifications of the substrates or by temperature gradients (Marangoni effect) at the solid walls, have already found many applications in microfluidic actuation. The flow induced surface tension effects are examined on the basis of a sharp interface model. It is demonstrated how nanoscale objects placed at the boundary of the flow domain result in generation of substantial surface forces acting on the bulk flow. The effect, studied in general for arbitrary two-dimensional obstacles, is shown to be the strongest for a lattice of one-dimensional threads.

5:19PM LJ.00009 Electromagnetically-driven capillary switches and oscillators , BERNARD MALOUIN, AMIR HIRSA, Rensselaer Polytechnic Institute, MICHAEL VOGEL — By designing pinned-contact, coupled droplet pairs at the appropriate length scale to promote surface tension as the dominant force, one can create bi-stable capillary switches and natural oscillators. These systems have been triggered by pressure pulses, electrochemistry, and electroosmosis. These methods are typically accompanied by bulky setups or slow response times. An alternate approach exploits electromagnetic activation. Our device consists of a millimeter scale orifice, overfilled with an aqueous ferrofluid, in proximity to a wire coil that generates a magnetic field. Experimental evidence of such capillary switches and energy efficient oscillators is presented here. Comparisons to a simplified model are also presented. This activation method is shown to have relatively fast response times, low driving voltages, and individual addressability. Electromagnetically activated capillary switches and oscillators offer many applications ranging from high-speed adaptive optics to micro-actuators, with possible circuit board integration.

5:32PM LJ.00010 Dynamics of pinned-contact oscillating gas/liquid lenses , AMIR HIRSA, BERNARD MALOUIN, JOSEPH OLLES, Rensselaer Polytechnic Institute, CARLOS LOPEZ, Intel Corp., MICHAEL VOGEL, Cornell University — Liquid lenses are a natural solution for applications in adaptive optics requiring a fast response. Existing liquid lenses use large disturbances to overcome liquid inertia and subsequently utilize the lens after the oscillations have dampened. An alternate strategy was demonstrated recently: a harmonically-driven liquid lens with an oscillating focal length that allows the capture of any plane in a given range by grabbing the image 'in sync' with the oscillations. Hence, by continuously oscillating the lens, the task of changing the focal length is effectively transformed from a mechanical manipulation to the electronic timing of image capture by the sensor, which can be achieved much faster. Good optical quality is possible by designing the liquid lens to traverse states where capillarity produces spherical interfaces. Energy efficiency is achieved through pinned contact lines and operation at resonance. We present experimental results along with predictions for the dynamics of such oscillatory driven lenses, including the effects of liquid volume, driving frequency and amplitude on droplet shapes and resultant optical characteristics. High fidelity imaging was demonstrated at 100 Hz for a millimeter scale liquid lens.

Monday, November 23, 2009 3:35PM - 5:45PM –

Session LK Multiphase Flows V 101J

3:35PM LK.00001 An efficient and accurate coupling between Lagrangian front-tracking and unstructured Eulerian grids , XIAOYI LI, MARIOS SOTERIOU, MARCO ARIENTI, United Technologies Research Center — Evolution of flow of immiscible fluids under high-shear poses severe challenges to the development of accurate and robust numerical techniques that can maintain a sharp separating interface. The implicit volume of fluid (VOF) advection using High Resolution Interface Capturing (HRIC) scheme offers the advantage of numerical stability at large time steps, but has been observed to cause interface diffusion at high shear. Advection using the standard Piecewise Linear Interface Calculation (PLIC), on the other hand, requires much smaller time steps. We have developed an efficient, accurate coupling approach between a sharp-interface front-tracking library and an *unstructured-grid* implicit flow solver. The high efficiency results from a localized searching algorithm for grid cells close to the interface. The accuracy is obtained from a conservative interfacial force transfer between front and grid that preserves momentum balance and from a novel approach for constructing the density profile across the interface. Validation of the method with tests of drop deformation in high shear will be presented, with attention to efficiency and accuracy. The performance of this stand-alone front-tracking capability on unstructured grids suggests that the coupled approach may be well suited for simulations in complex domains.

3:48PM LK.00002 Sharp Interface Cartesian Grid Method for High Speed Multi-material Dynamics Problems , SHIV KUMAR SAMBASIVAN, H.S. UDAYKUMAR, The University of Iowa — The dynamic response of materials to high-speed and high-intensity loading conditions due to shock waves, detonation waves, high-velocity impact and penetration processes is important in several applications including high-speed flows with droplets, bubbles and particles, and hypervelocity impact and penetration. To simulate such complicated high-pressure physics problems,a fixed Cartesian grid approach in conjunction with level set interface tracking is attractive. In this work, a sharp interface, Cartesian grid-based, Ghost Fluid Method is developed for resolving embedded fluid, elasto-plastic solid and rigid objects in hyper-velocity impact and high-intensity shock loaded environment. The embedded multi-material interface is tracked and represented by virtue of the level set interface tracking technique. The evolving interface and the flow are coupled via the GFM approach by meticulously enforcing the boundary conditions and jump relations exactly at the interface. A reflective boundary condition based approach is used to enforce the conditions on the interface. The subcell position of the interface and the topology of the interface are carefully embedded in the interpolation procedure. In addition, a tree-based Local Mesh Refinement scheme is employed to efficiently resolve the desired physics. The broad range of results presented in this work demonstrates the flexibility and robustness of the current approach.

4:01PM LK.00003 ABSTRACT WITHDRAWN –

4:14PM LK.00004 A level set based method for modeling large density ratio, interfacial flows , MEHDI RAESSI, HEINZ PITSCH, Center for Turbulence Research, Stanford University — We present a numerical methodology in the context of the level set method for modeling interfacial flows characterized by large density ratio. In this method, to advect momentum the conservative form of the momentum equation is solved. Using the level set function, the density of momentum fluxes is calculated based on the interface evolution. The same flux density is used for advecting mass, and thereby a tight coupling between mass and momentum transport is established. We present a set of results in which the density ratio ranges from 650 to 10,000 and demonstrate the capability of the method in handling flows with large interface deformations.

4:27PM LK.00005 An Embedded Boundary Method for Solving the Elliptic And Parabolic Interface Problems and Its application to the Stefan Problem , SHUQIANG WANG, ROMAN SAMULYAK, JAMES GLIMM, XIAOLIN LI, Stony Brook University — The embedded boundary method (EBM) on a Cartesian grid developed by Johansen and Colella has been extended to solve the elliptic/parabolic problems in 2D with an interior boundary (also called elliptic interface, parabolic interface problem). The method is a finite volume method. 2nd order accuracy in L_∞ norm is achieved. As its application, a Stefan problem is solved. Problems with multiple components (3 or 4 components) meeting at a single Cartesian cell was also solved by using this method. The algorithm is implemented in C++. The computational domain is partitioned using Cartesian grid. To reduce the computational memory needed, each cell of the Cartesian grid could change itself to be different type for cells with at most 2 components or more complex cells with 3 or 4 components. Test results for 3D elliptic interface problems also show that it is 2nd order accurate in L_∞ norm.

4:40PM LK.00006 Dynamics of morphology formation in phase-separation fronts , ALEXANDER WAGNER, ERIC FOARD, Department of Physics, North Dakota State University — We discuss the formation of domains in the wake of a phase-separation front in one, two and three dimensions. Perhaps surprisingly, if such a phase-separation front migrates with a constant velocity it will form very regular structures of lamellar domains or cylindrical columns oriented either parallel or orthogonal to the phase-separation front. We show analytical predictions of how the structure, orientation, and size of the formed domains depends on the speed of the phase-separation front (as well as the volume fraction) if the dynamics is purely diffusive and show numerical verification of our theoretical results. When hydrodynamics becomes important, however, the results are more complicated: in more than one dimension the formed structures become more fractal in nature, with structures on many different length-scales.

4:53PM LK.00007 Dynamics of particle clusters at fluid/fluid interfaces[1] , SRINATH MADHAVAN[2], Ph.D. Candidate, University of Alberta, PETER MINEV[3], Professor, University of Alberta, KRISHNASWAMY NANDAKUMAR[4], Professor Emeritus, University of Alberta — This talk is oriented toward research that describes the hydrodynamics of dense (relative to the lower fluid in a gravitational field) rigid particles at fluid-fluid interfaces through Direct Numerical Simulations (DNS). Understanding the factors that control the formation and stability of the complex rag layer (typically encountered during oil-water separation) is a motivation for the current study. The fundamental aspects of the problem at hand bear a connection with the formation of tight clusters of floating particles. Strong capillary forces are thought to promote this behavior. One of the challenges toward realizing the same in a numerical simulation is the implementation of a physically realistic boundary condition for the three phase moving contact line (MCL). To this end, we implement the recently proposed continuum form of the Generalized Navier Boundary Condition (Gerbeau and Lelievre, 2009) in a levelset and fictitious-domain based finite-element scheme and demonstrate its usefulness and accuracy through case studies.

[1]Support from the Discovery grant of the National Science and Engineering Research Council of Canada (NSERC) is gratefully acknowledged.

[2]Department of Chemical and Materials Engineering

[3]Department of Mathematical and Statistical Sciences

[4]Currently Cain Chair at the Cain Department of Chemical Engineering at Louisiana State University

5:06PM LK.00008 Theoretical base and numerical tools for modeling transitions between continuous and disperse multiphase motions[1] , DUAN ZHANG, XIA MA, PAUL GIGUERE, Los Alamos National Laboratory — Transitions between continuous and disperse multiphase motions happen commonly in nature and in our daily life. The phenomena include dissolving sugar cubes in a cup, formation of rain and hail, shattering a piece of glass. The capability of numerically simulating these phenomena is both important to industrial applications and to the understanding of nature. Relative to other aspects in this topic, theories for disperse multiphase flow is better developed despite many important issues still to be resolved. The theory for continuous multiphase flow is still in its infancy. The study of transition between continuous and disperse multiphase motion is at an even earlier stage of development. In this talk, we describe a possible theoretical framework based on the probability and statistical theory and a useful numerical method in simulating these phenomena. Deficiencies in the theory and in the numerical method are also discussed.

[1]Work performed under the auspices of the United States Department of Energy.

5:19PM LK.00009 Role of liquid Weber and Reynolds number in the primary breakup of turbulent liquid jets in crossflow , MADHUSUDAN PAI, Stanford University, OLIVIER DESJARDINS, University of Colorado, HEINZ PITSCH, Stanford University — Atomization of liquid fuel controls the combustion efficiency and pollutant emissions from internal combustion engines and gas turbines. A liquid jet injected into a crossflow breaks up by developing liquid surface instabilities and deformations due to aerodynamic sources and liquid jet turbulence, among other causes. There is a pressing need to understand the origin and role of these instabilities in the breakup of a liquid jet. These instabilities can be accurately quantified in detailed numerical simulations of liquid jets. A spectrally-refined interface (SRI) tracking scheme for interface transport coupled to an accurate and robust Navier-Stokes/Ghost-fluid method gas-phase solver is employed to perform large-scale detailed numerical simulations of liquid jets in a laminar crossflow. The liquid Weber number controls the tendency of a liquid jet to break up, while the liquid Reynolds number controls the range of length scales in the liquid jet turbulence. The interplay and role of these phenomena in the primary breakup of liquid jets is quantified through a parametric study. Existing models for turbulent primary breakup of liquid jets in crossflow are reviewed based on the numerical results.

5:32PM LK.00010 An identification of coherent structures in an inhomogeneous turbulence , RYUICHI NAGAOSA, RISS, AIST — This study proposes a proper scaling of a vortex indicator for practical identification of coherent structures (CS) from an inhomogeneous wall turbulence at the Reynolds numbers of $Re_\tau = 180$ and 400. The Laplacian of the pressure, Θ, is scaled by its time-space average Θ_{ave} and root-mean-square value Θ_{rms} to obtain a proper scaling for effective identification of CS, $\hat{\Theta} = (\Theta - \Theta_{ave}) / \Theta_{rms}$. Numerical turbulent flow realizations with a wall and a gas-liquid interface obtained by a direct numerical simulation technique are employed to confirm suitability of the proposed scaling. The results of this study exhibit that the probability density function (PDF) of $\hat{\Theta}$ is very similar without respect to the distance from the wall, unlike PDF of unscaled Θ. Because of this statistical similarity, identification of CS based on $\hat{\Theta}$ is shown advantageous to separate all the essential CS from disorganized turbulent background using a unique threshold level, especially in the region adjacent to the wall and the gas-liquid interface. Several turbulent flow signatures at the gas-liquid interface suggest that a threshold of $1 \leq \hat{\Theta} \leq 2$ is preferable to identify all the essential CS in the whole of the turbulent flow domain, without "contamination" caused by misidentification of meaningless structures.

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LL CFD IV: Multiphase Flows 200A

3:35PM LL.00001 Gradient-Augmented Level Set and Sub-Grid Accuracy in Multi-Phase Simulations , JEAN-CHRISTOPHE NAVE, MIT — In this presentation we will discuss a new method for solving the advection equation for a level set function. The approach relies on carrying both function values and gradients of the level set function as coupled evolved quantities, using Hermite interpolants and a semi-Lagrangian strategy. Some benefits compared to the traditional WENO approach include: better mass conservation, no need to solve a reinitialization equation, ability to capture features smaller than the grid resolution, computational speed, optimally local stencils, and second order accurate curvature calculation. To demonstrate the value of the new method, we will present several two-phase flow simulations for which sub-grid accuracy is important.

3:48PM LL.00002 A finite element method for simulating thermally fluctuating Brownian particles: Random force vs Random stress[1] , UMA BALAKRISHNAN, T.N. SWAMINATHAN, R. RADHAKRISHNAN, D.M. ECKMANN, P.S. AYYASWAMY, University of Pennsylvania — Targeted nanocarrier drug delivery holds promise for personalized medicine, but its optimization requires an accurate description of carrier motion. Our computational approach is aimed at situations where both Brownian motion and the hydrodynamic interactions are important. We consider two ways of assessing this motion: (a) time-correlated random forces (colored noise) acting on the particles, (b) hydrodynamic random stresses in the fluid equations (white noise). The first approach is geometry dependent, the number of random numbers (R_n) generated per step is equal to the number of particles. However, the second approach is geometry independent and requires R_n to be equal to the number of mesh points. Both the approaches have been investigated and the results are validated by comparing the calculated temperature of the system, with that determined from the equipartition theorem, and by comparing the predicted mean square displacement with that calculated from Einstein's formula. While both approaches yield comparable accuracies, the random stress approach appears to be more robust and readily generalizable to complex particle as well as flow geometries.

[1]Supported by: NIH-R01EB006818

4:01PM LL.00003 ABSTRACT WITHDRAWN –

4:14PM LL.00004 Computational Fluid Dynamics Study of the Effect of Turbulence and Two-Phase Flow on Flow Blurring Atomization[1] , JORGE RAMON, Florida International University, WILLARD SCHREIBER, University of Alabama — A novel atomization mechanism known as Flow Blurring (FB) mixes air with liquid to produce a fine spray. While the geometry of Flow Blurring is simple, the fluid mechanics of the two-phase mixing is complicated. CFD modeling of the Flow Blurring injector has been attempted previously assuming laminar one phase mixing between two different density gases. The objective of the present work was to study the effect of adding turbulence and two-phase flow to the previous CFD model. The k-ε, k-ω, and Reynolds stress models were investigated for representing turbulence. The k-ε realizable model produces the best results both from the standpoint of physical realism and numerical convergence and allows the Reynolds number based on flow characteristics of the FB injector to be increased by a factor of six. Three models of two-phase flow were examined: Volume of Fluid, Mixture, and Eulerian, none of which satisfactorily simulated two-phase mixing in the FB atomizer.

[1]Work performed under REU site sponsored by NSF grant EEC0754117.

4:27PM LL.00005 Steady/Unsteady Solutions of Full 2-D Governing Equations of Internal Condensing Flows, Their Responses to Flow Disturbances, and Their Controllability through Exit Conditions , SHANTANU KULKARNI, AMITABH NARAIN, SOUMYA MITRA, Michigan Technological University — This paper presents novel computational results obtained for the full 2-D governing equations of condensing flows in a channel (shear driven or gravity driven). It is shown that the internal condensing flows can be operated under two different boundary conditions at the exit, viz. unspecified exit condition and specified exit condition. These computational results state that for unspecified exit conditions, there exists a unique steady solution for the condensing flows, termed as "natural" solution. This "natural" solution is obtained by solving "strictly" steady governing equations. This paper demonstrates that the unsteady equations of condensing flows are *elliptic* and for the specified exit conditions different than the "natural" exit condition, one obtains an unsteady or quasi-steady solution based on the type of exit condition control. This paper demonstrates different ways to control the exit condition as well as solution attainability limits for gravity and shear driven flows.

4:40PM LL.00006 An Eulerian Numerical Method for Fluid-Solid Interaction , YANG ZHANG, MIT ERL, KEN KAMRIN, Harvard SEAS, JEAN-CHRISTOPHE NAVE, MIT MATH — Fluid-solid interaction is a difficult computational problem, primarily because solids and fluids are described in different perspectives — solid laws are written in Lagrangian frame while fluids are represented in Eulerian. Our work attempts to resolve this dilemma using a new method for Eulerian solid mechanics. We study the interaction of a large-deformation elastic solid with a Newtonian fluid in a single computational framework. We use a level set to track the interface between the two phases. The standard projection method is used to impose incompressibility in both phases, and the equations are discretized with an explicit, staggered finite-difference scheme. In the current implementation, a smeared Heaviside function is used to blur material properties across the interface. Simulations of various test cases will be presented in this talk.

4:53PM LL.00007 Efficient Simulation of Fully Coupled Wave-Body Interactions on a Cartesian Grid[1] , JIANMING YANG, FREDERICK STERN, University of Iowa — The sharp interface Cartesian grid method by Yang and Stern (Sharp interface immersed-boundary/level-set method for wave-body interactions, J. Comput. Phys. 228 (2009) 6590-6616) is extended for the efficient simulation of fully coupled wave-body interactions of a single body with a two-phase incompressible flow. The overall approach is based on a fractional-step method using finite differencing on a staggered Cartesian grid and the governing equations are solved in a non-inertial reference frame following the motion of the body. A level set method is adopted for the fluid-fluid interface tracking and a direct forcing immersed boundary method is used for the fluid-solid boundary treatment. The relationship between the body velocity and the explicit momentum forcing given by Kim and Choi (Immersed boundary method for flow around an arbitrarily moving body, J. Comput. Phys. 212 (2006) 662-680) is utilized for the non-iterative solution of fluid-structure interactions. The forces and moments calculation in the framework of solid-liquid-gas system is emphasized. Several cases ranging from water entry problems to ship moving in waves are demonstrated to showcase the accuracy and efficiency of the method.

[1]Sponsored by the Office of Naval Research (ONR) under grants N00014-01-1-0073 and N00014-06-1-0420.

5:06PM LL.00008 Numerical simulation of the formation of liquid bridge with movable contact line by an external electric field , JIN SEOK HONG, IN SEOK KANG, Chemical Engineering, Pohang University of Science and Technology, Korea — The authors have suggested a new dispensing method of droplets on demand, which is a counter-electrode-free electrohydrodynamic method with no pump and an inverted geometry; a top substrate and a bottom nozzle. Using this method, the authors have also demonstrated highly uniform dispensing results with a variance 1.8 % in droplet diameter. The dispensing process consists of two stages of liquid bridge (LB) formation by an applied electric pulse and its break-up by the movement of top plate. In this work, numerical simulation is performed for the first LB formation stage. The dynamics of liquid surface during LB formation is analyzed numerically for a simple liquid such as water. Especially, the movement of upper contact line is studied with respect to the dynamic contact angle model for the top plate. In addition, the asymmetry of the contact line mobility between nozzle and the top plate is also considered and its effect on the final upper contact area is analyzed. The numerical results are compared with the experimental results and discussed in terms of the effect of applied electric pulse.

5:19PM LL.00009 Numerical simulation of the flow patterns within concentric spheres with rotation and solid-liquid phase change , ARES CABELLO, RUBEN AVILA, Engineering Faculty — We present the flow patterns in the interior of a spherical annulus at different Taylor (Ta) numbers and for two Stefan (St) numbers. The equations of the two phases are solved in a Cartesian coordinate system using a spectral element method, in a reference frame that is turning with the system, then the centrifugal and Coriolis terms are considered. Firstly the Stefan number was equal to zero i.e. without solidification from the outer boundary. Secondly the St number was fixed to $St = 6 \times 10^{-3}$, hence the growing of a solid crust from the outer sphere was allowed. When the Ta number is in the range $Ta < 3 \times 10^5$ (subcritical regime), it is observed a basic flow. A transitional oscillatory stage appears when the Ta number is increased in the interval ($3 \times 10^5 < Ta < 2 \times 10^6$). When the Ta number is increased furthermore $Ta > 2 \times 10^6$ (supercritical regime) the oscillation of the flow dissapears and a new basic flow pattern is attained with four lobes located adjacent to the equator of the spheres. The influence of the solidification from the outer sphere on the flow, has been investigated by fixing the St number to $St = 6 \times 10^{-3}$ and for Ta number Ta =3.6$\times 10^6$. It is observed that as the solid crust grows, the Ta number dynamically decreases however since the viscous characteristic time of the fluid is higher than the rate of crust growth, the flow pattern is not modified.

5:32PM LL.00010 An Improved Bubble Packing Method for Unstructured Mesh Generation with Applications in Computational Fluid Dynamics , LILONG WU, BIN CHEN, State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an, China — An improved Bubble Packing Method (BPM) is proposed to generate high-quality unstructured mesh for prediction of the flow dynamics in a domain with complex geometries. For curved-boundary domain, firstly each curved boundary is mapped into a straight line, and then new bubble positions will be mapped into curved boundary back by arc-length parameterization method after the bubble system on the mapped straight line reaches equilibrium. In this way the bubble's departure from curved boundaries during dynamic movement of the bubbles can be avoided. Moreover, the grid density can be controlled simply and effectively. Local mesh refinement is realized by adding different size bubbles to the real / artificial vertices of the domain and bubble information of these vertices is transferred to the inner nodes of the domain by using the Shepard interpolation method. In order to validate the proposed algorithm, unstructured collocated grid systems are developed to numerically simulate lid-driven flow in square and polar cavities. The good agreement between numerical simulations with literatures under different Reynolds numbers confirms the effectiveness and feasibility of our proposed algorithm.

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LM Supersonic/Hypersonic II 200B

3:35PM LM.00001 Shock generated vorticity in the interstellar medium and the origin of the stellar initial mass function , NICHOLAS KEVLAHAN, RALPH PUDRITZ, McMaster University — Observations of the interstellar medium (ISM) and molecular clouds suggest these astrophysical flows are strongly turbulent. The main observational evidence for turbulence is the power-law energy spectrum for velocity fluctuations, $E(k) \propto k^{\alpha}$, with $\alpha \in [-1.5, -2.6]$. The Kolmogorov scaling exponent, $\alpha = -5/3$, is typical. At the same time, the observed probability distribution function (PDF) of gas densities in both the ISM as well as in molecular clouds is a log-normal distribution. In this paper we examine the density and velocity structure of interstellar gas traversed by curved shock waves in the kinematic limit. We demonstrate mathematically that just a few passages of curved shock waves generically produces a log-normal density PDF. This explains the ubiquity of the log-normal PDF in many different numerical simulations. We also show that subsequent interaction with a spherical blast wave generates a power-law density distribution at high densities, qualitatively similar to the Salpeter power-law for the IMF. Finally, we show that a focused shock produces a *downstream* flow with energy spectrum exponent $\alpha = -2$. Subsequent shock passages reduce this slope, achieving $\alpha \approx -5/3$ after a few passages. These results suggest that fully-developed turbulence may *not* be required to explain the observed energy spectrum and density PDF.

3:48PM LM.00002 Surface Catalysis Modeling of Air-SiO2 Systems Under Hypersonic Conditions Using ReaxFF MD Simulation , PAUL NORMAN, TOM SCHWARTZENTRUBER, University of Minnesota, IOANA COZMUTA, NASA Ames — The high-speed entry of a blunt body into Earth's atmosphere brings about the dissociation of diatomic nitrogen and oxygen molecules via the shockwave formed in front of the body. Through surface catalysis, these dissociated atoms can recombine on the heat shield of the body, increasing its overall heating. The goal of this project is to study surface catalysis on amorphous silicon-dioxide (SiO2), a significant component in the reusable thermal protection system used on the Space Shuttle. Specifically, our objective is to determine the rates of recombination of monatomic N and O for the range of temperatures and pressures experienced by a heat shield during Earth re-entry. Additionally, we aim to determine the rates of specific reaction mechanisms on a SiO2 surface, including adsorption, desorption, surface diffusion, and various recombination processes. This is accomplished by performing large reactive molecular dynamics simulations using the ReaxFF force field, which naturally allows bond formation/breaking to occur during the course of a molecular dynamics simulation. Several methods for speeding up the equilibration and collection of rates for low-pressure gas-surface systems (typical of re-entry conditions) where events become infrequent will also be discussed.

4:01PM LM.00003 A Tightly Coupled Solver for Hypersonic Ablation Problems , NATHAN MULLENIX, ALEX POVITSKY, University of Akron Dept. of Mechanical Engineering — Ablation is a process of rapid material removal from a solid surface by chemical reactions, sublimation and other erosive processes, absorbing large quantities of heat, and is one of the techniques used for thermal protection on hypersonic vehicles. It consists of several coupled sub-processes including gas dynamics, heat transfer, and ablative mechanisms at the surface. The past state of the art models include only a subset of these and generally ignore transient phenomena involving shape changes (i.e. formation of cavities). The current study presents the development of a solution methodology for the ablation problem in which the model for each sub-process are linked at the point of their development, the solution of each is tightly coupled to the solution of the others, and shape changes effects are intrinsically included. Starting from first principles, the Reynolds Transport Theorem is used to derive a set of governing equations that takes into account the movement of the ablating surface and the resulting mass transfer. Existing explicit-in-time finite-volume numerical methods are modified for this set, and a reactive-Riemann solver is derived for ablative fluxes. Methods for avoiding numerical artifacts such as plumes of ablated material are described. Results are provided for graphite ablation in hypersonic flow, and are compared to relevant experiments, and their sensitivity to particular parameters will also be presented.

4:14PM LM.00004 ABSTRACT WITHDRAWN –

4:27PM LM.00005 Chemistry-Vibration Coupling in CO_2 system for High Enthalpy Nozzle Flows , SRIRAM DORAISWAMY, University of Minnesota - Twin Cities, DANIEL KELLEY, University of Missouri -St. Louis, GRAHAM CANDLER, University of Minnesota - Twin Cities — The present work investigates the complex process of vibrational relaxation and its subsequent coupling with the chemical processes in high enthalpy nozzle flows. High enthalpy CO_2 nozzle expansion in reflected shock tunnels shows significant difference in shock standoff distance between computational and experimental results. CO_2 being a linear triatomic molecule has three modes of vibration - bending, symmetric stretch and antisymmetric stretch modes. To better model the vibrational relaxation, the bending and the symmetric stretch modes were coupled into one mode due the fact that these modes are strongly coupled through Fermi resonance. Furthermore, to simplify the analysis, this coupled mode was assumed to be in equilibrium with the translational mode. For the CO_2 only the antisymmetric mode is considered. A vibrational state-specific model was devised by considering the first few vibrational states of the diatomic species in a CO_2 system, i.e. CO_2, CO and O_2. The rate constants for the vibrational relaxation processes were obtained from experimental data. This vibrational model is then coupled with a chemistry model to run the full flowfield nozzle simulation, and also to obtain the shock standoff distance.

4:40PM LM.00006 Comparison of CFD and Theoretical Post-Shock Gradients in Hypersonic Flow[1] , GRAHAM CANDLER, University of Minnesota — In recent work of Hornung, expressions for the gradients of flow properties immediately behind a curved shock wave were obtained for a reacting gas. In this work, I use the expressions derived by Hornung to compare with inviscid computational fluid dynamics simulations of a Mach 8 flow over a cylinder. A finite-rate vibrational relaxation model is used to simplify the comparisons with theory. The shape of the bow shock wave is extracted from the CFD results, fitted with a polynomial, and then used to compute the post-shock gradients of the main flow variables. It is found that in general the CFD results are in very good agreement with the theory for both perfect gas and vibrationally relaxing flows. There are some notable differences, mostly centered on the location of the change in sign of the post-shock density gradient; this quantity is found to be very sensitive to the relaxation rate of the gas. The theoretical post-shock gradients provide a rigorous test of CFD and suggest possible experiments that would be very sensitive test of the models of finite-rate vibrational and chemical processes.

[1]This work was sponsored by the Air Force Office of Scientific Research under grant FA9550-04-1-0341 and by the Department of Defense National Security Science and Engineering Faculty Fellowship.

4:53PM LM.00007 Accelerated Molecular Dynamics Simulation of Hypersonic Flow Features in Dilute Gases , THOMAS SCHWARTZENTRUBER, PAOLO VALENTINI, University of Minnesota — Accurate simulation of high-altitude hypersonic flows requires advanced physical models capable of predicting the transfer of energy between translational, rotational, vibrational, and chemical modes of a gas in strong thermochemical non-equilibrium. A combined Event-Driven / Time-Driven (ED/TD) Molecular Dynamics (MD) algorithm is presented that greatly accelerates the MD simulation of dilute gases. The goal of this research is to utilize advances in computational chemistry to study thermochemical non-equilibrium processes in hypersonic flows. The ED/TD MD method identifies impending collisions (including multi-body collisions) and advances molecules directly to their next interaction, however, then integrates each interaction accurately using an arbitrary interatomic potential via conventional MD with small timesteps. First, the ED/TD MD algorithm and efficiency will be detailed. Next, ED/TD MD simulations of normal shock waves in dilute argon will be validated with experiment and direct simulation Monte Carlo simulations employing the variable-hard-sphere collision model. Profiling of the code reveals that the relative computational time required for the MD integration of collisions is extremely low and the potential for incorporating advanced classical and first-principles interatomic potentials within the ED/TD MD method will be discussed.

5:06PM LM.00008 On the Structure of Plasma Liners for Plasma Jet Induced Magneto Inertial Fusion , LINGLING WU, ROMAN SAMULYAK, Stony Brook University — 3D simulations of the formation and evolution of plasma liners for the Plasma Jet Induced Magneto Inertial Fusion (PJMIF) have been performed. In the PJMIF concept, a plasma liner, formed by merging of a large number of radial, highly supersonic plasma jets, implodes on the target in the form of two compact plasma toroids, and compresses it to conditions of the nuclear fusion ignition. The propagation of a single jet with Mach number 60 from the plasma gun to the merging point was studied using the front tracking code FronTier. The simulation result was used as input to the jet merger problem. The merger of 144 jets and the formation and heating of plasma liner by oblique shock waves was studied and compared with recent theoretical predictions. The main result of the study is the prediction of the average Mach number reduction and the description of the liner structure and properties.

5:19PM LM.00009 Coupled computational fluid-thermal investigation of hypersonic flow over a quilted dome surface[1] , CHRISTOPHER OSTOICH, DANIEL BODONY, PHILIPPE GEUBELLE, University of Illinois at Urbana-Champaign — The hypersonic environment is characterized by the high temperatures that are generated in the fluid at a vehicle surface. In the effort to enable the operation of lightweight, reusable hypersonic vehicles, flexible, thin thermal protection panels have been considered to mitigate thermal loads. High surface temperatures create through-the-thickness thermal gradients which cause the panels to bow, resulting in changes to the external flow field and leading to a fully coupled fluid-thermal-structural problem. Certain aspects of the fluid-thermal (no structural) coupling were examined in a 1980s NASA Langley experiment of a Mach 5.74 laminar boundary past an array of spherical domes. We reexamine this case computationally using a high-fidelity Navier-Stokes solver coupled with a thermal solver to investigate the effects on the flow and resulting heat load on the structure due to the bowed panels. Specifically the surface temperature, surface heat flux, and downstream boundary developments are reported, and compared with experiment.

[1]Supported by the U.S. Air Force Research Laboratory Air Vehicles Directorate under contract number FA8650-06-2-3620.

5:32PM LM.00010 Penetrator Nose Drag Measurements in Supersonic Flows[1] , JOSEPH HOLLAND[2], PHILLIP SCHINETSKY, YESENIA TANNER, SEMIH OLCMEN[3], STANLEY JONES[4], University of Alabama — In the current study, a rigid body penetrator nose shape that is optimized for minimum penetration drag (Jones et al., 1998) has been tested to determine the aerodynamic drag of such a penetrator in comparison to three additional nose shapes. Other nose shapes tested were an ogive cylinder, a 3/4 power series nose, and a standard cone. Fineness ratio for the studied nose geometries was chosen as l/d = 1 to maximize variation of the aerodynamic drag forces acting on the nose shapes. The experiments were carried out in the University of Alabama's 6" x 6" supersonic wind tunnel, using a 4 component force balance system. Each of the nose shapes were tested at nine different Mach numbers ranging from 1.99 to 3.65. Results show that the nose shape optimized for penetration has the lowest drag coefficient of all the shapes at each Mach number within an uncertainty of 5.75 %.

[1]Work performed under REU site sponsored by NSF grant EEC 07554117
[2]Undergraduate Research Assistant
[3]Associate Professor
[4]University Research Professor

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LN Non-Newtonian Flows III 200C

3:35PM LN.00001 Blistering pattern and formation of nano-fibers in capillary thinning of polymer solutions , CHRISTIAN WAGNER, RAINER SATTLER, STEPHAN GIER, Technische Physik, Universitaet des Saarlandes, 66041 Saarbruecken, JENS EGGERS, School of Mathematics, University of Bristol, University Walk, Bristol BS8 1TW — When a dilute polymer solution experiences capillary thinning, it forms an almost uniformly cylindrical thread, which we study experimentally. In the last stages of thinning, when polymers have become fully stretched, the filament becomes prone to instabilities, of which we describe two: A "breathing" instability, originating from the edge of the filament, and a sinusoidal instability in the interior, which ultimately gives rise to a Rayleigh Plateau instability followed "blistering" pattern of beads on the filament. We describe the linear instability with a spatial resolution of 80 nm in the disturbance amplitude. Preliminary micro-PIV measurements indicate the existence of irregular flow fields. For sufficiently high polymer concentrations, the filament eventually separates out into a "solid" phase of entangled polymers, connected by fluid beads. A solid polymer fiber of about 100 nanometer thickness remains, which is essentially permanent.

3:48PM LN.00002 Spatially-dependent elastic instabilities in flow around an array of cylinders , LICHAO PAN, BARRY SCHARFMAN, PAULO ARRATIA, University of Pennsylvania — When flexible polymer molecules (in dilute solution) flow around a cylinder, they are strongly stretched due to the combination of both curved streamlines and extensional flow. Here, the flow of a viscoelastic fluid around an array of cylinders is investigated in a microchannel. As the strain- rate is varied at low Reynolds number ($<10^{-2}$), tracer and particle-tracking experiments show that molecular stretching produces two flow instabilities, one in which the velocity field becomes asymmetric, and a second in which it fluctuates non-periodically in time. These instabilities are spatially-dependent in the sense that the two instabilities may be present at a single values of strain-rate (or Wissenberg number) but at different locations in the microchannel.

4:01PM LN.00003 Dynamical instability of viscoelastic fluids driven by steady roll-mills , BIN LIU, MICHAEL SHELLEY, Courant Institute, New York University, JUN ZHANG, Department of Physics and Courant Institute, New York University — A layer of viscoelastic fluid, made of a polymeric solution, is driven from beneath by 16 rotating disks – rollers – on a square lattice. Each adjacent pair of rollers rotate in opposite directions at constant speed. We focus on the region near the free-surface of the fluid, where the dynamics is roughly two-dimensional. A set of stagnation points are thus created between the rollers, and divides the driven fluid into 16 cells. When the strain rate due to the local flow geometry is small compared to the relaxation time of the fluid, the fluid behaves like a Newtonian one, giving rise to fluid cells of equal size located above each individual roller. As the forcing increases, symmetries are broken, and the cells start differentiating in size. We observe experimentally that when the forcing is great enough, the asymmetric flow pattern becomes unsteady, and the stagnation points oscillate spontaneously. We show that the oscillatory frequency depends on the Weissenberg number Wi, characterizing the ratio of the forcing time scale to the relaxation time of the fluid.

4:14PM LN.00004 A Transition to Mixing and Oscillations in a Stokesian Viscoelastic Flow , BECCA THOMASES, Dept. of Mathematics, UC-Davis, MICHAEL SHELLEY, Courant Institute, New York University — To understand observations of low Reynolds number mixing and flow transitions in viscoelastic fluids, we study numerically the dynamics of the Oldroyd-B viscoelastic fluid model. The fluid is driven by a simple time-independent forcing that creates a cellular flow with extensional stagnation points. We find that at O(1) Weissenberg number these flows lose their slaving to the forcing geometry of the background force, become oscillatory with multiple frequencies, and show continual formation and destruction of small-scale vortices. This drives flow mixing. These new flow states are dominated by a single large vortex, which may be stationary or move persistently from cell to cell. Increasing the number of degrees of freedom by increasing the number of driving cells broadens the temporal frequency spectrum and improves fluid mixing.

4:27PM LN.00005 Thermal instabilities in melt spinning of viscoelastic fibers , CHUNFENG ZHOU, SATISH KUMAR, University of Minnesota — Nonisothermal melt spinning of viscoelastic fibers where the viscosity varies in a step-like manner with respect to temperature is studied in this work. A set of one-dimensional equations based on the slender-jet approximation and the upper-convected Maxwell model is used to describe the melt spinning process. The process is characterized by the force required to pull the fiber, the strength of external heating, and the draw ratio, the square of the ratio of the fiber diameter at the spinneret to that at the take-up roller. For low levels of elasticity and sufficiently strong external heating, there can be three pulling forces consistent with the same draw ratio, similar to the Newtonian case studied by Wylie et al. (J. Fluid Mech. 570 (2007) 1-16). For higher levels of elasticity, the process exhibits a draw ratio plateau where the draw ratio hardly changes with the pulling force, reflecting a competition between thermal and elastic effects. As in the Newtonian case, external heating introduces a new instability—termed thermal instability—that is absent in isothermal systems. Linear stability analysis reveals that external heating improves stability for low levels of elasticity, but can worsen stability for higher levels of elasticity, which is again a consequence of the interplay between thermal and elastic effects. The results of the present work demonstrate a possible mechanism through which external heating can stabilize the melt spinning of viscoelastic fibers.

4:40PM LN.00006 Interfacial instability of turbulent two-phase stratified flow with non-Newtonian rheology , LENNON O'NARAIGH, PETER SPELT, Imperial College London — We study the stability of a stratified flow configuration where the bottom layer exhibits non-Newtonian rheology, and where the top layer is Newtonian, fully developed, and turbulent. We first derive a base-state model to describe the equilibrium flow in the flat-interface state, which takes into account the yield stress and power-law nature of the bottom fluid, while a closure model is used to constitute the Reynolds stresses in the upper fluid. Next, we develop a linear-stability analysis to predict when the base state is unstable, and pay particular attention to characterizing the influence of the non-Newtonian rheology on the instability. Increasing the yield stress (up to the point where unyielded regions form in the bottom layer) is destabilizing; increasing the flow index, while bringing a broader spectrum of modes into play, is stabilizing. In addition, a second mode of instability is found, which depends on conditions in the bottom layer. For shear-thinning fluids, this second mode becomes more unstable, and yet more bottom-layer modes can become unstable for a suitable reduction in the flow index. One further difference between the Newtonian and non-Newtonian cases is the development of unyielded regions in the bottom layer, as the linear wave on the interface grows in time. These unyielded regions form in the trough of the wave, and can be observed in the linear analysis for a suitable parameter choice.

4:53PM LN.00007 Channel Flow of Wormlike Micellar Solutions , MICHAEL CROMER, PAM COOK, University of Delaware, GARETH MCKINLEY, Massachusetts Institute of Technology — We examine the inhomogeneous response of the VCM model (Vasquez, Cook, McKinley 2006) in steady pressure-driven channel flow. The VCM model, a microstructural network model, was developed to describe concentrated solutions of wormlike micelles. The model comprises of a set of coupled partial differential equations, which incorporate breakage and reforming of two micellar species (a long species 'A' and a shorter species 'B') in addition to reptative and Rousian stress-relaxation mechanisms. We examine pressure-driven flow in microfluidic devices with rectangular cross-sections as well as with hyperbolic converging/diverging walls. The velocity profile predicted by the VCM model in Poiseuille flow deviates from the parabolic profile expected for a constant viscosity fluid and exhibits strong shear bands near channel walls. This shear-banding is analogous to that seen in circular Taylor-Couette flow and in good qualitative agreement with experimental observations in microfluidic channels. The hyperbolic planar contraction is of special interest due to the dominant contribution of extensional flow along the centerline and the proposed use of such flows as microfluidic extensional rheometers. The model predictions are compared with birefringence measurements of the evolution in the local microstructural orientation of CTAB and CPyCl-based micellar solutions.

5:06PM LN.00008 Instability of the air cavity of a micellar solution in the wake of a submerged rod , THOMAS OBER, GARETH MCKINLEY, Dept of Mechanical Eng, MIT, SUNGHWAN JUNG, Dept. of Math, MIT — The behavior of flowing surfactant and polymeric solutions is of increasing importance as these materials are used more commonly as rheological modifiers. Here, we investigate the instability of the air cavity formed in the wake of a rod, which is submerged in the oncoming stream of a non-Newtonian fluid with a free surface exposed to air. Two fluid systems with different concentrations of cetylpyridinium chloride (CPyCl), sodium salicylate (NaSal) and NaCl are studied. Under certain conditions, the cavity exhibits a repeating tooth-like pattern, whose wavelength and amplitude vary with depth, rod diameter, oncoming velocity and fluid properties. We characterize experimentally the cavity closing dynamics, and the wavelength and amplitude along the profile of the cavity, probing the interplay between viscoelasticity, inertia and hydrostatic pressure in our experiments. Finally, we propose a simple model to capture the some of the key features of the dynamics of the air cavity.

5:19PM LN.00009 Field-controlled adhesion in confined magnetorheological fluids , JOSE MIRANDA, SERGIO LIRA, Departamento de Fisica, UFPE, Brazil — The study of reversible, functional, and controllable adhesives is a matter of considerable practical interest, and academic research. We report the adhesive response of a magnetorheological fluid confined between two parallel plates under a probe-tack test, when it is subjected to an applied magnetic field. Our analytical approach is based on a Darcy-like law formulation which considers a magnetic field-dependent yield stress behavior. The adhesion force is calculated in closed-form for two different configurations produced by a Helmholtz coils setup: uniform perpendicular, and nonuniform radial magnetic fields. In both cases, we verify that adhesion force is hugely increased as a result of the field-dependent nature of the yield stress. This provides a versatile way to obtain a shear resistant, tough structural adhesive through magnetic means.

5:32PM LN.00010 Magnetohydrodynamic channel flow with Braginskii's anistropic viscosities[1] , PAUL DELLAR, University of Oxford — We study the channel flow of a fluid obeying Braginskii's magnetohydrodynamics, in which the viscosity parallel to the magnetic field lines greatly exceeds the viscosity in perpendicular directions. This reflects a weakly collisional regime where particles interact primarily through coupling to the magnetic field, rather than directly through inter-particle collisions. Contrary to the conclusion of a recent study, there is no well-defined limit as the ratio of perpendicular to parallel viscosities tends to zero. The maximum velocity grows as the minus one-quarter power of the (small) viscosity ratio $(\mu_\perp/\mu_\parallel)^{-1/4}$, due to large shears that develop across boundary layers at the walls. The width of these boundary layers scales as the three-quarters power of the viscosity ratio. They thus lie inside analogs of the usual Hartmann layers, and the Lorentz force does not enter their leading-order force balance. The long-time behavior of computations using lattice Boltzmann magnetohydrodynamics, which is readily adapted for anisotropic viscosities, is in excellent agreement with these asymptotic solutions.

[1]Supported by EPSRC ARF grant EP/E054625/1.

Monday, November 23, 2009 3:35PM - 5:19PM –
Session LP Couette Flow 200D

3:35PM LP.00001 Bi-Stable Turbulent Spherical Couette , DANIEL S. ZIMMERMAN, SANTIAGO ANDRÉS TRIANA, DANIEL P. LATHROP, University of Maryland College Park — We study the turbulent shear flow between differentially rotating concentric spheres of radius ratio $\eta = r_\mathrm{i}/r_\mathrm{o} = 0.35$ in the University of Maryland three meter spherical Couette device. We impart rapid overall rotation and reach a region of parameter space of Ekman number $\nu/\Omega_\mathrm{o}(r_\mathrm{o} - r_\mathrm{i})^2$ of $10^{-6} < E < 10^{-7}$ with Rossby number $\Delta\Omega/\Omega_\mathrm{o}$ of $0.05 < Ro < 5$. We present direct flow measurement data including wall shear, wall pressure, local flow velocity and driving torques. We observe a rich variety of phenomena including inertial wave excitation and bi-stablity. The flow in bi-stable parameter regions exhibits spontaneous transitions between global mean flow states on time scales very long compared to the typical fluctuation time scales. This has interesting consequences for angular momentum transport including non-monotonicity of the driving torques with increasing differential rotation rate.

3:48PM LP.00002 Turbulent Taylor-Couette flow between independently rotating cylinders , MATTHEW S. PAOLETTI, DANIEL P. LATHROP, University of Maryland — We present experimental studies of the turbulent flow of water between independently rotating cylinders. The Taylor-Couette system is capable of both strong turbulence ($Re > 10^6$) and rapid rotation (Ek $< 10^{-7}$). The torque required to drive the inner cylinder is precisely measured as a function of the two angular velocities Ω_i and Ω_o. Of particular interest are three distinct regions of the (Ω_i, Ω_o) parameter space defined by the inner and outer boundaries having equal: (i) angular velocities (solid-body rotation), (ii) azimuthal velocities and (iii) angular momenta (Rayleigh criterion) with the outer boundary stationary line (Ω_o= 0) serving as the final bound. We supplement the global torque measurements with local wall shear stress measurements as a means of detecting Coriolis-restored, linear inertial modes. We model the system as being composed of two interacting, turbulent boundary layers. There are several open questions that we hope to be able to answer: (1) Are there conditions under which angular momentum will flow uphill? (2) What quantity (angular velocity, azimuthal velocity, or angular momentum) does the system most effectively "mix," and does that depend upon system parameters.

4:01PM LP.00003 Precession in a laboratory model of the Earth's core , SANTIAGO TRIANA, DANIEL ZIMMERMAN, Department of Physics and IREAP, University of Maryland, DANIEL LATHROP, Department of Physics, IREAP and IPST, University of Maryland — The Earth's rotation axis precesses with a period of 25800 years, caused mainly by the combined torques of the sun and the moon acting on the slight equatorial bulge of the planet. Without precession (or convection), a viscous core will come to rotate as a solid body with the mantle. The fluid core responds to the precessional forcing and the resulting motion can in principle power the geodynamo. There have been several attempts, both theoretical and experimental, to unveil the role of precession in the motion of the fluid core. A three meter diameter spherical-Couette system with a 1m inner sphere is the most recent and largest experimental model of the Earth's core. The experiment provides data at parameter ranges much closer to the Earth's compared to what was possible before either experimentally or numerically. Experimental data from this system evidencing precessional forcing will be presented and compared to theoretical predictions.

4:14PM LP.00004 Spin-up and Spin-down in a Spherical Annulus , MATTHEW ADAMS, SANTIAGO TRIANA, DANIEL ZIMMERMAN, DANIEL LATHROP, University of Maryland, College Park — We present experimental studies of turbulent fluid flow in a spherical annulus with approximate radius ratio 1/3 that is spun up and spun down. One experimental apparatus uses water as the working fluid, and provides localized measurements of velocity, wall shear, and pressure. The other experimental apparatus has sodium as the working fluid, and uses magnetic field measurements to extract information about the global flow within the device. The geometry of the experiments makes these studies potentially applicable to geophysical and astrophysical bodies. In particular, Mercury is known to have an at least partially fluid core, and during its orbit its rotation rate increases and decreases periodically. Preliminary results of spin-down indicate a transfer of energy to the so-called spin-over mode as the turbulence decays after the outer and inner spheres have been brought to a stop.

4:27PM LP.00005 Incipient spots in plane Couette flow , BRUNO ECKHARDT, Philipps-Universitaet Marburg, TOBIAS M. SCHNEIDER, Harvard University, DANIEL MARINC, RWTH Aachen — We investigate by direct numerical simulations the transition from laminar to turbulent plane Couette flow in long and wide domains. Previous studies in small domains have established that the boundary between laminar and turbulent is formed by the stable manifold of a persistent structure, called the edge state. Because of the small size of the domain and its periodic continuation in downstream and spanwise directions, this edge state was infinitely extended. Using an adaptation of the edge state tracking algorithm to larger domains we could detect edge states that are localized in the spanwise, in the downstream or in both directions. The structures are dominated by downstream vortices, and they are found to be exponentially localized in the downstream direction, and faster than exponentially in the spanwise direction. The structures serve as nuclei for the formation of spots and provide estimates for optimal widths and intensities. They evolve towards space filling turbulence by first increasing in energy content then by spreading in the surrounding laminar regions.

4:40PM LP.00006 Spatially localized solutions and homoclinic snaking in plane Couette flow , JOHN GIBSON, Georgia Institute of Technology, TOBIAS SCHNEIDER, Harvard University, JOHN BURKE, Boston University — We examine a new class of spatially localized solutions to plane Couette flow, first discovered by Schneider, Marinc, and Eckhardt. Under continuation in Reynolds number the equilibrium and traveling-wave solutions exhibit a sequence of saddle-node bifurcations strikingly similar to the "homoclinic snaking" phenomenon observed in simpler PDE systems such as the Swift-Hohenberg equation. These localized solutions originate from bifurcations off the spatially periodic equilibria discovered by Nagata and others and retain their physical structure, demonstrating the relevance of exact periodic solutions to turbulent flows in spatially extended domains.

4:53PM LP.00007 Twente Turbulent Taylor-Couette , DENNIS VAN GILS, CHAO SUN, DETLEF LOHSE, University of Twente, Physics of Fluids, The Netherlands, PHYSICS OF FLUIDS TEAM — A newly constructed turbulent Taylor-Couette (TC) system consists of two independently rotating cylinders of 0.93 m in length and the inner and maximum outer radii are respectively 0.20 m and 0.28 m. The maximum rotation rates are 1200 RPM for the inner cylinder and 600 RPM for the outer cylinder. With the working fluid held at a constant temperature within at least 0.1 degree Celsius the setup allows for precisely controlled measurements. The maximum Reynolds number that can be achieved with inner cylinder rotation only, is estimated to be around 2×10^6. This allows for measurements well into the turbulent regime. The system is designed not only for single-phase flow studies, but also for two-phase flow research like bubbly drag reduction. The clear acrylic outer cylinder and several view ports in the top and bottom plate allow for optical measurement techniques such as LDA, PIV and PTV. Instead of only measuring global quantities like drag, temperature and pressure, the system is also equipped with a multitude of sensors for measuring local quantities. Combining the detailed localized information from inside of the gap with the global quantities, provides a way to study the mechanisms of bubbly drag reduction.

5:06PM LP.00008 Drag and lift forces on a counter-rotating cylinder in rotating shear flow , CHAO SUN, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, the Netherlands, TOM MULLIN, Manchester Centre for Nonlinear Dynamics, University of Manchester, United Kingdom, LEEN VAN WIJNGAARDEN, DETLEF LOHSE, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, the Netherlands — We experimentally investigated the motion of a heavy cylinder in a drum filled with water, and rotating about a horizontal axis. The cylinder either co-rotates or counter-rotates with the rotating drum. The flow field around the cylinder, both for co-rotation and counter-rotation situations, was measured with Particle Image Velocimetry in order to investigate the different flow mechanism. For the counter-rotation situation, the cylinder freely rotates without contact with the wall of the drum, due to the lift force acting on it. The drag and lift coefficients, on the freely counter-rotating cylinder, were measured in a wide range of Reynolds numbers $2{,}500 < \mathrm{Re} < 25{,}000$ and dimensionless rotation rates $0.0 < \alpha < 1.2$. We found that the drag coefficient is consistent with previous measurements on a cylinder in a uniform flow. However, a significant enhancement of the lift coefficient is observed in the present measurements. We expect the enhancement of the lift force is caused by the combined effects of rotation of the cylinder and the vicinity of a wall.

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LQ Microfluidics: General 200E

3:35PM LQ.00001 Numerical Investigation of Liquid Flow through Micro-channels with Post Patterned Super-hydrophobic Walls , A. AMIN, D. MAYNES, B.W. WEBB, Brigham Young University — We numerically investigate the effect of post patterned super-hydrophobic surfaces on the drag reduction for laminar liquid flow through micro-channels. Hydrophobic surfaces exhibiting micro-scale structures can significantly reduce the liquid-solid contact area resulting in reduced surface friction. The effects of cavity fraction (the ratio of cavity area to total surface area) and relative module width (ratio of post/cavity repeating length to channel hydraulic diameter) on the slip-length and on the Darcy friction factor-Reynolds number product, *fRe*, were explored numerically for the post structured hydrophobic walls. The cavity fraction and relative module width vary from 0.5 to 0.9998 and from 0.01 to 1.5, respectively. In general, as both cavity fraction and relative module width increase *fRe* decreases. The present results are compared with those for surfaces exhibiting rib/cavity patterns that are parallel and perpendicular to the flow direction. At high cavity fractions the post/cavity structuring produces larger slip-length and greater reduction in *fRe* than either parallel or perpendicular rib/cavity structures. The results are also compared with scaling laws previously published in the literature.

3:48PM LQ.00002 A universal scaling for viscous flows around microfabricated pillars , NIMISHA SRIVASTAVA, CARL MEINHART, UC Santa Barbara — Complex geometries that involve an intricate network of channels and pillars are increasingly being used in microfluidic devices. Central to the successful operation of these devices is a fundamental theoretical framework that explains the effects and interplay of viscous, inertial and capillary forces in these geometries. One such geometry is a dense (1000 by 1000) array of micron-sized pillars. We will present a universal scaling (over four orders of magnitude) that predicts viscous, pressure driven flows in these pillars. We have developed a finite element model using COMSOL Multiphysics to simulate Stokes flow between pillars. Using curve fitting on flow through a wide range of height, diameter and gap (an order of magnitude), we were able to derive a unique model that will accurately predict flow rates in any given random array of pillars. We have found that the pressure driven viscous flow within pillars depends almost linearly with the height (h) of the pillars while it varies inversely with the square root of the diameter (d) of the pillars. The flow rate follows a 2.33 power of the gap between the pillars. In addition, we have, *for the first time*, observed lubrication- like scaling in low Reynolds number (**Re**<0.5) viscous flows around an array of microfabricated pillars. Our experiments and simulations have explored and validated the design space when $\mathbf{h}/\mathbf{g}$ and $\mathbf{g}/\mathbf{d}$ is between 1 and 10 –s as is the case in most microfluidic applications, which makes this finding imperative for future design of geometries involving pillars, tortuous channels and porous structures.

4:01PM LQ.00003 A universal scaling for viscous flows around micro- and nano-fabricated pillars , CARL MEINHART, NIMISHA SRIVASTAVE, CHANGSONG DING, NOEL MACDONALD, UCSB — Complex geometries that involve an intricate network of channels and pillars are increasingly being used in microfluidic devices. Central to the successful operation of these devices is a fundamental theoretical framework that explains the effects and interplay of viscous, inertial and capillary forces in these geometries. One such geometry is a dense (1000 by 1000) array of micron-sized pillars. We will present a universal scaling (over four orders of magnitude) that predicts viscous, pressure driven flows in these pillars. We have developed a finite element model that simulates Stokes' flow between pillars. Building upon a universal scaling law for viscous losses in the pillars, we developed a model that accurately predicts flow rate through the pillars. We have found that pressure-driven viscous flow within the pillars depends nearly linearly with the height (h), inversely with the square root of the diameter (d), and a power law behavior with the gap between the pillars. In addition, we have observed lubrication- like scaling in low Reynolds number (Re <0.5) viscous flows around an array of microfabricated pillars. Numerical results compare well to experimental observation.

4:14PM LQ.00004 Bistability in a simple fluid network due to viscosity contrast , BRIAN STOREY, Olin College, JOHN GEDDES, Olin College and University of Aberdeen, Scotland, DAVID GARDNER, Olin College, RUSSELL CARR, University of New Hampshire — We study the existence of multiple equilibrium states in a simple fluid network using Newtonian fluids and laminar flow. We demonstrate theoretically the presence of hysteresis and bistability, and we confirm these predictions in an experiment using two miscible fluids of different viscosity; sucrose solution and water. Possible applications include bloodflow, microfluidics, and other network flows governed by similar principles.

4:27PM LQ.00005 Streaming Flow in Branching Micro/Mini Channels[1] , DONNA MEYER, ZONGQIN ZHANG, CHANG LIU, THOMAS BAREK, AHMED FADL, University of Rhode Island, MANFRED KRAFCZYK, Technical University of Braunschweig, Germany — A streaming-based study is presented in branching micro and minichannels of oscillating flows with no net mass flow. Zero-mean velocities result in distinct differences between forward and backward flow velocity profiles, causing near-wall particles in the fluid, and those near the channel center, to advance in opposite directions. Streaming velocities were found to be highly influenced by oscillating amplitude and frequency, as shown in numerical analyses and validated analytically. A kinematic viscosity of the fluid which is larger than the diffusivity of the particles were found to result in effective convective transport. Advantages of streaming flow-based phenomenon include enhanced mixing, pumpless fluid propulsion, multichannel fluid distribution, easy system integration with cost-effective operation. The distinguishing features of streaming flow lend themselves to numerous applications.

[1]The research was supported by the National Science Foundation, Grant No. OISE-0530203.

4:40PM LQ.00006 Microfluidics on a wire , TRISTAN GILET, DENIS TERWAGNE, NICOLAS VANDEWALLE, GRASP, ULg — In this talk, we discuss the behavior of droplets sliding on fibers. An on/off transition is observed when a droplet comes around an intersection between several fibers: large droplets cross the junction while small droplets remain blocked. We show that simple fiber networks may perform advantageously most operations of digital microfluidics, such as multiplexing: intersections could be the basic component of new fiber-based microfluidic devices.

4:53PM LQ.00007 Systems-level analysis of AC microfluidics and the problem of trapped gas bubbles , SØREN VEDEL, Dept. of Micro- and Nanotechnology, Technical University of Denmark, DTU Nanotech Bldg. 345 East, DK-2800 Kongens Lyngby, LAURITS HØJGAARD OLESEN, Device Research and Technology, Novo Nordisk A/S, Brennum Park, DK-3400 Hillerød, Denmark, HENRIK BRUUS, Dept. of Micro- and Nanotechnology, Technical University of Denmark, DTU Nanotech Bldg. 345 East, DK-2800 Kongens Lyngby, Denmark — Using pulsatile pressure and flow rate, we extend the equivalent circuit (EC) approach for systems-level analysis of microfluidic systems to also include dynamic, transient effects such as inertia and compliance. The dynamic time scales of microfluidics are typically on the order of millisecond, or equivalently frequencies in the low kHz regime. A novel pressure source has been developed and successfully tested for the experimental generation of flow under these conditions for two microfluidic setups. Good agreement was found between the experimental observations and the results of corresponding systems-level EC model [1]. Trapped air bubbles in the microfluidic system severely influences its performance, while also leading to erroneous predictions from the systems-level analysis. We present theoretical analysis of the physics of bubble adhesion to the system walls, leading to insights to their removal.
[1] S. Vedel, L.H. Olesen, H. Bruus, Lab Chip (submitted 2009), http://arxiv.org/abs/0907.2679

5:06PM LQ.00008 Convective Heat-Transfer Characteristics of Laminar Flow Through Smooth- and Rough-Wall Microchannels , V.K. NATRAJAN, K.T. CHRISTENSEN, University of Illinois, Urbana — The convective heat-transfer behavior of laminar flow through smooth- and rough-wall microchannels is investigated by performing *non-intrusive* measurements of fluid temperature using a microscale adaptation of two-color laser-induced fluorescent thermometry for flow through a heated copper microchannel testbed of hydraulic diameter $D_h = 600\,\mu$m. These measurements, in concert with pressure-drop measurements, are performed for a smooth-wall case and two different rough-wall cases with roughness that is reminiscent of the surface irregularities one might encounter due to imperfect fabrication methods. Pressure-drop measurements reveal the onset of transition above $\mathrm{Re_{cr}} = 1800$ for the smooth-wall case and deviation from laminar behavior at progressively lower Re with increasing surface roughness. The local Nusselt number (Nu) for smooth-wall flow over the range $200 \leq \mathrm{Re} \leq \mathrm{Re_{cr}}$ agree well with macroscale predictions in both the thermally-developing and -developed regimes. With increasing roughness, while an enhancement in local Nu is noted in the thermally-developing regime, these differences do not exist upon attainment of a thermally- developed state. Examination of temperature profiles across the microchannel suggest that the thermal boundary layer may be regenerated by roughness, resulting in a delay in the attainment of thermally-developed flow.

5:19PM LQ.00009 Optimization of Microchannel Characteristics for Enhanced Heat Transfer in the Laminar Regime , R. SAKSENA, K.T. CHRISTENSEN, Univ. of Illinois — The present effort explores optimization of microchannel characteristics for enhanced convective heat transfer in the laminar regime with specific application to the development of complex microfluidic networks for self-cooling material systems. Of particular interest is optimization of the layout of the microscale flow passages in a manner that both maximizes heat transfer while simultaneously minimizing any additional pressure-drop penalty compared to straight microchannels. This optimization is achieved using a multi-objective genetic algorithm in conjunction with numerical simulations in the flow regime relevant to self- cooling applications ($\mathrm{Re} < 100$) that span a parameter space of both wavelength and amplitude for sinuous microchannels. Experimental validation of the identified optimal configurations is performed in a controlled heat-transfer environment using microchannels written with a unique robocasting fugitive-ink-deposition printer at UIUC.

5:32PM LQ.00010 Hysteresis and wall-effects in low Reynolds number propulsion by driven elastic filaments , SARAH CLARK, PRABHAKAR RANGANATHAN, JAMES FRIEND, Monash University — There is currently intense interest in developing micron-sized robots for uses such as minimally invasive surgery. Although progress has been made in miniaturizing the motor, the hydrodynamic behavior of associated propellers is far from being fully understood. An example is an elastic filament driven by a torque at one end where the shape assumed by the filament is strongly coupled to the hydrodynamics forces. Investigation of these dynamics has only recently commenced, for instance Manghi et al. [PRL 96, 068101 (2006)] uncovered an intriguing shape transition in an elastic filament spun in a bulk fluid. Since such transitions can be expected to have a crucial bearing on the operation of microbot swimmers we examine this behavior in detail with simulations. We also study the effect of planar no-slip walls on the propulsion characteristics. The slender filament is represented as a bead-spring chain and inter-bead hydrodynamic-interactions are described using the appropriate Greens functions. We study the origin of the shape transition and hysteresis in detail and show the relationship to sedimenting filaments. We show that the presence of a boundary either perpendicular or parallel to the axis of the applied torque has a significant effect on the overall motion. We also point out the possible detrimental consequences of these effects on operation of microbots in the vicinity of conduit walls.

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LR Convection II 200F

3:35PM LR.00001 A Comparison of Turbulent Thermal Convection Between Conditions of Constant Temperature and Constant Flux: Simulation Methods , HANS JOHNSTON, University of Massachusetts - Amherst, CHARLIE DOERING, University of Michigan - Ann Arbor — We report the results of high resolution direct numerical simulations of two-dimensional Rayleigh-Bénard convection for Rayleigh numbers up to $Ra = 10^{10}$ in order to study the influence of temperature boundary conditions on turbulent heat transport. Specifically, we consider the extreme cases of fixed heat flux (where the top and bottom boundaries are poor thermal conductors) and fixed temperature (perfectly conducting boundaries). Both cases display identical heat transport at high Rayleigh numbers fitting a power law $\nu \approx 0.138 \times Ra^{.285}$ with a scaling exponent indistinguishable from $2/7 = 0.2857\ldots$ above $Ra = 10^7$. The overall flow dynamics for both scenarios, in particular the time averaged temperature profiles, are also indistinguishable at the highest Rayleigh numbers. The findings are compared and contrasted with results of recent three-dimensional simulations.

3:48PM LR.00002 A Comparison of Turbulent Thermal Convection Between Conditions of Constant Temperature and Constant Flux: Simulation Results[1] , CHARLES R. DOERING, University of Michigan, HANS JOHNSTON, University of Massachusetts — We report the results of high resolution direct numerical simulations of two-dimensional Rayleigh-Bénard convection for Rayleigh numbers up to $Ra = 10^{10}$ in order to study the influence of temperature boundary conditions on turbulent heat transport. Specifically, we considered the extremes of fixed heat flux (where the top and bottom boundaries are poor thermal conductors) and fixed temperature (perfectly conducting boundaries). Both cases display identical heat transport at high Rayleigh numbers fitting a power law $\nu \approx 0.138 \times Ra^{.285}$ with a scaling exponent indistinguishable from $2/7 = .2857\ldots$ above $Ra = 10^7$. The overall flow dynamics for both scenarios, in particular the time averaged temperature profiles, are also indistinguishable at the highest Rayleigh numbers.

[1]Supported by NSF Awards PHY0555324, SCREMS0619492, and PHY0855335.

4:01PM LR.00003 Effects of Shear on Unstably Stratified Convection , CURTIS HAMMAN, PARVIZ MOIN, Stanford University — Direct numerical simulations of plane channel flow heated from below are examined. At Richardson numbers of order unity ($|Ri| = Ra/PrRe^2$, $10^6 \leq Ra \leq 10^9$, $180 \leq Re_\tau \leq 590$ and $Pr = 1$), the production of wall-normal baroclinic vorticity is shown to have a substantial effect upon the mean momentum balance that leads to enhanced turbulent mixing, asymmetry, and turbulent heat transfer. These effects are neglected by traditional Boussinesq formulations, which generate zero baroclinic vorticity in the direction of gravity, accelerate negative and positive density fluctuations identically, and make no direct contribution to the streamwise mean momentum balance. Motivated by the work of Shirgaonkar and Lele (*Physics of Fluids*, 2006), a computationally efficient extension to the Boussinesq approximation is developed and shown to accurately capture these essentially incompressible interactions between shear and buoyancy as observed in variable-density turbulence. Applications to heat exchangers and other advanced energy systems, in which the effects of shear and buoyancy are comparable, are highlighted.

4:14PM LR.00004 ABSTRACT WITHDRAWN –

4:27PM LR.00005 Large-scale flows in free and mixed convection[1] , JORGE BAILON-CUBA, MOHAMMAD EMRAN, JOERG SCHUMACHER, Technische Universitaet Ilmenau — Convective turbulence in closed volumes is associated with large-scale circulations of the flow (LSC). They depend sensitively on the geometry and the physical parameters, such as Rayleigh and Prandtl numbers. Here, we consider two systems: free convection in cylindrical cells and mixed convection in a complex rectangular setting with local heat sources. The LSC and the amount of heat transferred is determined by the so-called proper orthogonal decomposition (POD) of the turbulent fields. We apply the so-called snapshot method to extract the modes from DNS data. The most energetic POD modes give us insight into the dynamic dominance of coherent flow and temperature patterns, and how well the original inhomogeneous flow can be modeled with a reduced number of modes in a low-dimensional model. For example, in case of the cylindrical cell the primary POD mode transfers about one half of the total amount of heat through the vessel. For the mixed convection case, the influence of the geometry and the inflow conditions on these LSC structures is also addressed.

[1]The work is supported by the Deutsche Forschungsgemeinschaft.

4:40PM LR.00006 The development of a buoyant vortex (thermal) in stationary and plane stagnation flows[1] , GALI ALON, JIMMY PHILIP, JACOB COHEN, Technion, Faculty of Aerospace Engineering — The evolution of a buoyant vortex (thermal) in stagnant and irrotational plane stagnation flows is studied using both computational and theoretical tools. The relative effect of external shear is explored through the ratio between the two relevant time scales associated with the shear and buoyancy (viscous and diffusive effects, although included, are considered to be relatively small). An important effect of the stagnation base-flow is the earlier penetration of the ambient fluid into the buoyant mass, causing the formation of a buoyant vortex ring. Consequently, the growth of circulation ceases earlier resulting in a lower value of the maximum circulation. The initial growth rate of the circulation is theoretically predicted to be proportional to the density difference and the vertical extension along the structure's symmetry line. In an attempt to describe the time development of the circulation, a simple Lagrangian model is proposed, and its results agree with the numerical ones. Finally, theoretical analysis verified numerically, shows that the fluid impulse grows linearly or exponentially in stationary fluid or in plane stagnation flow, respectively.

[1]This research was supported by the Israeli Science Foundation under Grant No. 1247/06.

4:53PM LR.00007 Convectons in periodic and bounded domains , EDGAR KNOBLOCH, University of California at Berkeley, ISABEL MERCADER, ORIOL BATISTE, ARANTXA ALONSO, UPC, Barcelona, Spain — Numerical continuation is used to compute spatially localized convection in a binary fluid with no-slip laterally insulating boundary conditions and the results compared with the corresponding ones for periodic boundary conditions. The change in the boundary conditions produces a dramatic change in the snaking bifurcation diagram that describes the organization of localized states with periodic boundary conditions: the snaking branches turn continuously into a large amplitude state that resembles periodic convection with defects at the sidewalls. Odd parity convectons are more affected by the boundary conditions since the sidewalls suppress the horizontal pumping action that accompanies these states in spatially periodic domains [O. Batiste et al., J. Fluid Mech. 560, 149 (2006)].

5:06PM LR.00008 Bounds on the Nusselt Number for Marangoni Convection[1] , GEORGE HAGSTROM, University of Texas at Austin, CHARLIE R. DOERING, University of Michigan — We use the background method to prove rigorous upper bounds on the Nusselt number in terms of the Marangoni number in Marangoni convection. When the Prandtl number is infinite $Nu \leq .84Ma^{2/7}$. For finite Prandtl number we proved that $Nu \lesssim Ma^{1/2}$. We compare these to numerical simulations by Boeck and Thess that suggest that for real flows $Nu \lesssim Ma^{2/9}$. We also use the background method and non-variational techniques to improve the lower bound for the critical Marangoni number for energy stability of the conduction solution in the infinite Prandtl number case.

[1]Supported by NSF PHY-0555324 and PHY-0855335.

5:19PM LR.00009 A Model of Convective Taylor Columns in Rotating Rayleigh Benard Convection , IAN GROOMS, KEITH JULIEN, JEFFREY WEISS, University of Colorado, Boulder, EDGAR KNOBLOCH, University of California, Berkeley — Many real fluid flows are nearly incompressible and are influenced by both rotation and thermal forcing. Rotation tends to suppress variation along the axis of rotation, while strong thermal forcing often gives rise to thermal plumes that travel vertically (in the direction of gravity). When the axis of rotation and gravity are aligned, or nearly so, these effects can combine to produce long lived columnar structures which have been observed in laboratory and numerical experiments; these "Convective Taylor Columns" can be interpreted as the effective particles "convectons" of the flow, accounting for a significant proportion of the vertical heat and momentum flux in the fluid and for the enhanced lateral mixing otherwise absent in non-rotating flows. However, due to the experimental challenges of 3-D data acquisition and the numerical challenges of simulation at low Rossby number, these structures remain poorly understood. We here present a nonlinear model for these columnar structures in the context of rotating Rayleigh-Bénard convection; our model makes use of multiscale asymptotics, complex variables, and special functions.

5:32PM LR.00010 ABSTRACT WITHDRAWN –

Monday, November 23, 2009 3:35PM - 5:45PM –

Session LS Waves I 200G

3:35PM LS.00001 Resonantly Forced Gravity–Capillary Lumps on Deep Water[1] , T.R. AKYLAS, YEUNWOO CHO, MIT — A theoretical study is made of the wave disturbance generated by a locally confined external pressure on the surface of deep water moving with speed V near the minimum gravity–capillary phase speed, c_{min}. According to linear inviscid theory, the response when V coincides with c_{min} is unbounded, and the interplay of nonlinear and damping effects is crucial close to this resonance. The analysis is based on an approximate model that combines the linear dispersion relation in the vicinity of c_{min} with quadratic and cubic nonlinearity as well as viscous damping. For V well below c_{min}, the transient response from rest approaches the small-amplitude steady state predicted by linear theory, but nonlinear effects come into play at a certain forcing speed, $c_{crit} < V$. Past this critical speed, that depends on the strength of the forcing, the response jumps to a finite-amplitude state comprising a gravity-capillary lump on the downstream side of the excitation, and a time periodic state is also possible for a range of forcing speeds slightly below c_{min}. This latter state involves periodic shedding of lumps that get damped quickly as they propagate downstream of the forcing. The theoretical predictions show good qualitative agreement with laboratory experiments conducted by J. H. Duncan and J. Diorio at U. of Maryland.

[1]Supported by NSF and AFOSR.

3:48PM LS.00002 An Experimental Investigation of the Wave Pattern Generated by a Moving Pressure Source: Solitary Capillary-Gravity Waves[1] , J.H. DUNCAN, J.D. DIORIO, A. LISIEWSKI, R. HARRIS, University of Maryland — The wave pattern generated by a small pressure source moving across a water surface at speeds less than the minimum phase speed for linear gravity-capillary waves (c_{min} = 23 cm/s) was investigated experimentally. The resulting wave pattern was measured using cinematic shadowgraph and laser-induced fluorescence (LIF) techniques. The results show the existence of several distinct behavioral states. At low speeds, no wave behavior is observed and the pattern resembles the symmetric stationary condition. However, at a critical speed, but still below c_{min}, the pattern undergoes a sudden transition to an asymmetric state with a stationary, 2D solitary wave that forms behind the pressure source. This solitary wave is elongated in the cross-stream relative to the stream-wise direction and resembles gravity-capillary "lumps" observed in previous numerical calculations. As the translation speed approaches c_{min}, another time-dependent behavior is observed characterized by periodic "shedding" from a V-shaped solitary wave pattern. This work will be discussed in conjunction with the recent numerical calculations of T. Akylas and his research group.

[1]Supported by NSF, Division of Ocean Sciences (OCE751853) and the ARCS Foundation.

4:01PM LS.00003 Internal Waves in Shear Flow , SCOTT WUNSCH, ALAN BRANDT, Johns Hopkins University Applied Physics Laboratory — Internal waves propagating through a shear flow can exchange energy with the mean flow. Waves may be reflected or transmitted through the shear, gaining or losing energy by exchange with the mean flow field. This effect is most pronounced at a "critical level," a depth where the wave horizontal phase velocity matches the local mean flow speed. Laboratory experiments are underway to study internal waves interacting with shear using the synthetic schlieren measurement technique. Particular interest is on conditions leading to internal wave amplification, which based on theory should occur when the Richardson number is less than 1/4. Experimental results indicate that, as the stratified shear flow is subject to instabilities as Richardson number approaches 1/4, internal wave interactions are more complex than the idealized theories predict.

4:14PM LS.00004 Propagation of internal waves through time-dependent shear profiles in the ocean and atmosphere , JULIE VANDERHOFF, Brigham Young University — Internal wave breaking in the ocean and atmosphere leads to mixing of pollutants and nutrients, contributes to the global mixing budget, and is necessary for the overall global circulation. Locations and magnitudes of this breaking are not fully understood. Internal waves are constantly being generated throughout the ocean and atmosphere. These waves can propagate long distances before breaking and dissipating. They will interact with other flow phenomenon as they propagate, including strong shear profiles, inertial scale waves, and vortex dipoles, as examples of a few. These interactions may lead to breaking, change the physical parameters of the short wave (including wavenumber and amplitude), change their direction of propagation (turning points), or shift them hundreds of kilometers from their original path. Ray tracing and numerical simulations are used to better understand the three-dimensional dynamics of these types of interactions. Results will be compared to previous experimental work and ocean observations from the Hawaiian Ocean Mixing Experiment. The results of these interactions further our knowledge of the evolution of the energy spectrum and give insight into how locally generated internal waves can contribute to the global energy budget.

4:27PM LS.00005 Internal wave structure emitted by a horizontally oscillating sphere[1] , EVGUENYI ERMANYUK, Lavrentyev Institute of Hydrodynamics (LIH), JAN-BERT FLOR, BRUNO VOISIN, Laboratoire des Ecoulements Geophysiques et Industriels (LEGI) — An oscillating body in a stratified fluid generates a double cone-shaped internal-wave pattern, the 3D analogue of the classic St.Andrew-cross. For sufficiently low frequency and large amplitude oscillations, higher-order wave harmonics may be generated along with the fundamental one. We present an experimental study of the 3D structure of first- and second-order wave fields emitted by a *horizontally* oscillating sphere. In contrast to the axisymmetric wave pattern found for a *vertically* oscillating sphere, for *horizontal* oscillations, the first- and higher-order-harmonic waves have different distributions of wave amplitudes in the azimuthal direction. The amplitude of the first-order waves is shown to follow the cosine dependence on the azimuthal angle, in accordance with theoretical predictions. The azimuthal distribution of the amplitude of the second-order waves gives evidence of a quadrupolar distribution, with four preferential directions of wave radiation in a horizontal plane, along the direction of oscillation and normal to it. Noteworthy is that the amplitudes of these second-order waves may exceed the amplitude of first-order waves.

[1]EE acknowledges the UJF (LEGI) for a visiting fellowship.

4:40PM LS.00006 A Model for Large-Amplitude Internal Solitary Waves with Trapped Cores , KARL HELFRICH, Woods Hole Oceanographic Institution, BRIAN WHITE, Univeristy of North Carolina at Chapel Hill — Large-amplitude internal solitary waves in continuously stratified systems can be found by solution of the Dubreil-Jacotin-Long (DJL) equation. For finite ambient density gradients at the surface (bottom) for waves of depression (elevation) these solutions may develop recirculating cores for wave amplitudes above a critical value. These recirculating cores contain densities outside the ambient range, may be statically unstable, and thus are physically questionable. To address these issues the problem for trapped-core solitary waves is reformulated. A finite core of uniform density and velocity, but unknown shape, is assumed. The core density is arbitrary, but generally set equal to the ambient density on the streamline bounding the core. The uniform core velocity set to the wave phase speed. The exterior flow satisfies the DJL equation and pressure continuity is imposed at the core boundary. Simultaneous numerical solution of the DJL equation and the core condition gives the exterior flow and the core shape. Numerical solutions of time-dependent nonhydrostatic equations with the theoretical solutions as initial conditions show that the waves are stable up to a critical amplitude above which shear instability destroys the initial wave. Trapped-core waves formed by lock-release initial conditions also agree well with the theoretical wave properties despite differences in the core circulation.

4:53PM LS.00007 The Cross-Stream Structure of the Crests of Breaking Waves[1] , J.D. DIORIO, X. LIU, J.H. DUNCAN, University of Maryland — The cross-stream profiles of spilling breaking waves (wavelengths 80-120 cm) are investigated experimentally. A programmable wave maker is used to generate Froude scaled wave packets (central frequencies 1.15 - 1.40 Hz and various wave maker amplitudes) that create breakers via dispersive focusing. A cinematic 2D LIF technique is used to measure the crest profile histories both in stream-wise and cross-stream planes. It is found that the cross-stream averaged amplitude undergoes periodic oscillations due to the passage of large streamwise (oriented parallel to the wave crest) ripples. Cross-stream ripples, while initially small, grow rapidly as breaking develops. These cross-stream ripples are in the range of 1-4 cm in wavelength and can have amplitudes comparable in size to the streamwise ripples. The amplitude of the cross-stream ripples grows with the gravity wavelength to the third power and shows periodic peaks that coincide with the troughs of the streamwise ripples. The cross-stream surface gradients show thin persistent surface "scars" that appear to be generated in the troughs of the streamwise ripples. The connection between these observations and a possible vortical model is discussed.

[1]Supported by NSF, Division of Ocean Sciences (OCE751853) and the ARCS Foundation.

5:06PM LS.00008 Air Bubble Entrainment by Breaking Ship Bow Waves[1] , M. TAVAKOLINEJAD, M. SHAKERI, D. HADSCHIEV, J.H. DUNCAN, University of Maryland — Air entrainment induced by breaking bow waves simulated with a 2D+T technique was studied experimentally in a tank that is 14.8m long, 1.15m wide, and 2.2m deep with a water depth of 1.85m. In the 2D+T technique, a two-dimensional wave maker moves horizontally and deforms in a manner that approximates the time varying intersection of one side of the hull of the three-dimensional ship and a fixed vertical plane oriented normal to the ship's path. The experiments were performed in simulated seawater and the bubble sizes and velocities in the streamwise plane were measured with a double-pulsed shadowgraph technique. In cases with plunging breakers, the primary mechanisms for air entrainment are the entrapment of a tube of air in the crest during the impact of the plunging jet and the turbulent fluid motion in the splash region create by the plunging jet impact. Two equivalent ship model forward speeds, one with a very weak plunging breaker and one with a very strong plunging breaker, were studied. Bubble size distributions, void fractions and bubble velocity distributions at times corresponding to the passage of the ship stern will be presented.

[1]This work is supported by the Office of Naval Research under grant number N000140810372. Scientific Officer Dr. P. Purtell.

5:19PM LS.00009 Experimental observation of trapped modes in a water wave channel , PABLO COBELLI, Laboratoire de Physique et Mécanique des Milieux Hétérogènes - PMMH - ESPCI, VINCENT PAGNEUX, Laboratoire d'Acoustique - Université du Maine - LAUM, AGNÈS MAUREL, Laboratoire Ondes et Acoustique - LOA - ESPCI, PHILIPPE PETITJEANS, Laboratoire de Physique et Mécanique des Milieux Hétérogènes - PMMH - ESPCI — The fluid around a free surface piercing circular cylinder in a long narrow wave tank can exhibit a local oscillation that does not propagate down the channel but is confined to the vicinity of the cylinder. This is a manifestation of the so-called trapped modes, bound states occurring in a wide variety of situations in physics. In this study, we present the first whole-field time resolved measurements for the free surface deformation obtained by a Fourier transform profilometry technique. The scattering characteristics of the cylinder and consequently the behavior of the trapped mode frequency are determined. The experimental results show good agreement with the predictions arising from linear water-wave theory.

5:32PM LS.00010 2D plus time analogy of corner waves downstream partially submerged bodies , PABLO MARTINEZ-LEGAZPI, JAVIER RODRIGUEZ-RODRIGUEZ, Carlos III University of Madrid, JUAN LASHERAS, University of California, San Diego — We have studied experimentally and numerically the expansion flow developing downstream the corner of a partially submerged vertical plate. In this flow configuration, a steady wave remains attached to the corner of the plate. Theoretical analysis shows that, taking advantage of the slender nature of the flow, the 3D steady problem can be transformed into a 2D+time one that resembles some important features of deep-water breaking waves. The resulting simplified problem is then solved using a boundary element method. Finally, the results of simulations are compared with experimental measurements.

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LT Suspensions I 200H

3:35PM LT.00001 Near-wall colloidal dynamics probed by evanescent-wave dynamic light scattering[1] , J. BLAWZDZIEWICZ, Yale University, E. WAJNRYB, IPPT, Warsaw, Poland, P. LANG, IFF, Juelich, Germany, Y.-N. YOUNG, NJIT, J.K.G. DHONT, IFF, Juelich, Germany, B. CICHOCKI, Warsaw University, Poland — We present theoretical, numerical, and experimental investigations of evanescent-wave dynamic light scattering (EWDLS) in a wall-bounded colloidal suspension of spheres. The first cummulant Γ_1 representing the initial decay of the time autocorrelation function of the scattered field is expressed in terms of the hydrodynamic tensor $\mathbf{H}_w(\kappa, \mathbf{q})$ describing response of the suspension to a spatially varying harmonic force damped exponentially away from the wall. The wavelength of the harmonic spatial variation corresponds to the scattering vector $\mathbf{q}$ in the EWDLS experiments, and the exponential decay is characterized by the decay length κ^{-1} of the evanescent wave. The hydrodynamic tensor $\mathbf{H}_w$ is evaluated using viral expansion at low densities and numerical simulations at higher densities. A complex non-isotropic structure of the tensor $\mathbf{H}_w(\kappa, \mathbf{q})$ reflects the hydrodynamic particle-wall coupling and wall-induced short range suspension ordering. Our theory and simulations agree well with the results of EWDLS experiments.

[1]Supported by NSF CAREER grant CBET–0348175.

3:48PM LT.00002 A 3-D Multiphase Particle Lattice Boltzmann Model of Colloidal Drop Dynamics , ABHIJIT JOSHI, YING SUN, Drexel University — A three-dimensional lattice Boltzmann method (LBM) has been developed for multiphase flows with particle suspensions. The unique challenge of this approach is to simulate the dynamics of liquid, vapor, and suspended particles in a stable manner. Adhesive forces between the suspended particles and the surrounding fluids are added to the previous single-phase particle suspension models and inter-particle forces are also taken into account. The model is first used to study the dynamics of colloidal drop coalescence for different particle sizes and concentrations. Results show that the liquid-vapor interface corresponds to a local energy minimum for the suspended particles. The wetting, dewetting, contact line pinning, and particle self-assembly of a colloidal drop is then examined as the drop spreads and evaporates on patterned (hydrophobic and hydrophilic) substrates. Evaporation is modeled using a quasi-static mass removal process. The drop wetting kinetics and final particle deposition are studied as a function of the surface energy step and pattern width of the substrate, as well as the particle size and volume fraction. Results are compared with experimental observations.

4:01PM LT.00003 Hydrodynamic interactions between two semi-flexible in-extensible filaments in Stokes flow[1] , YUAN-NAN YOUNG, New Jersey Institute of Technology — Hydrodynamic interactions between two semi-flexible in-extensible filaments are shown to have a significant impact on filament buckling and their subsequent motion in Stokesian fluids. In linear shear flow, hydrodynamic interactions lead to filament shear dispersion that depends on the filament aspect ratio and the initial filament separation. In linear extensional flow, hydrodynamic interactions lead to complex filament dynamics around the stagnation point. These results suggest that hydrodynamic interactions need to be taken into account to determine the self-diffusion of non-Brownian semi-flexible filaments in a cellular flow [Phys. Rev. Lett., 99, 058303, 2007].

[1]Support from NSF is acknowledged

4:14PM LT.00004 Field activated alignment and bifurcations in rigid rod suspensions , ARVIND GOPINATH[1], SARAH BATES[2], ANETTE HOSOI[3], MIT — Weak fields tend to align hard rod suspensions and trigger bifurcations from steady to periodic states. Three canonical problems are studied to thresh out common dynamical characteristics - the effect of weak shear, intrinsic aligning capabilities due to paramagnetic effects and externally imposed alignment via active orienting motors that adhere to the rods. The focus is on the relationships between new bifurcating solutions and the symmetries broken in the process of emergence. Imposition of weakly non-equilibrium shear flow can result in global bifurcations that are not discernable by a linear stability analysis while destabilization due to effects of polarity are seen to yield local bifurcations. Our exact results are corroborated by detailed asymptotic studies and extend previously obtained results.

[1]Mechanical Engineering, MIT.
[2]Mechanical Engineering, MIT
[3]Professor, Mechanical Engineering, MIT

4:27PM LT.00005 Numerical study of nonlinear interactions in suspensions of ideally polarizable spheres under electrophoresis , JAE SUNG PARK, DAVID SAINTILLAN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — We investigate the dynamics in a suspension of interacting ideally polarizable spheres undergoing electrophoresis using theory and numerical simulations. In addition to the classical linear electrophoretic motion of the particles, it is shown that two types of nonlinear electrokinetic phenomena may also occur and result in relative motions. First, when several particles are present and field gradients are created, nonzero dielectrophoretic forces (DEP) may arise due to Maxwell stresses in the fluid. In addition, if the particles can polarize under the action of the applied field, induced-charge electrophoresis (ICEP) occurs and may induce relative motions through hydrodynamic interactions. These two nonlinear effects are first analyzed in the prototypical case of two equal-sized spheres using asymptotic methods and are predicted to result in particle pairings. Based on this analysis, numerical simulations of full-scale particle suspensions undergoing DEP and ICEP with periodic boundary conditions are also performed using a fast Smooth Particle-Mesh Ewald (SPME) algorithm. The simulations confirm that pairing dynamics occur, and results are presented on the suspension microstructure, velocity statistics and particle hydrodynamic diffusion.

4:40PM LT.00006 The Polarization of Nanorods Submerged in an Electrolyte Solution and Subjected to an AC Electric Field[1] , HUI ZHAO, University of Nevada, HAIM BAU, University of Pennsylvania — Recently, there has been a growing interest in utilizing electric fields to position and separate rod-shaped particles such as DNA molecules, actin filaments, and nanorods. The polarization of the electric double layer enveloping the rod plays a critical role in determining the magnitude and direction of the rod's dipole moment. We calculate the dipole moment as a function of the electric field frequency, the rod's aspect ratio (length/radius), the rod's free surface charge, and the double layer's thickness. To this end, we solve the Poisson-Nernst-Planck equations for the ions' migration, diffusion, and convection. When the surface charge is small and the rod is short, the dipole moment is negative. As the rod's length increases, the dipole moment increases, and eventually changes sign from negative to positive. The dipole coefficient of rods, whose length is greater than some critical value, increases linearly with length. The theoretical predictions are compared and favorably agree with experimental data for short DNA molecules.

[1]The work was supported by the Penn Nano-Bio Interface Center (NSF NSEC DMR-0425780) and by NSF NIRT CBET-0609062.

4:53PM LT.00007 The Role of Particle Deformation in the Rheology and Microstructure of Noncolloidal Suspensions , JONATHAN CLAUSEN, DANIEL REASOR, CYRUS AIDUN, Georgia Institute of Technology — Particle deformation creates a marked effect on the rheology of noncolloidal suspensions. More pronounced non-Newtonian behavior such as shear-thinning and normal stress differences can be seen as compared with rigid particle suspensions. In this study, a lattice-Boltzmann-method fluid is coupled to a finite-element-method solid to simulate three-dimensional deformable particles. A Lees–Edwards boundary condition is implemented in the lattice-Boltzmann method, which allows the investigation of bulk suspension properties. Simulation results focus on shear viscosity and normal stress differences, as well as microstructure parameters such as the Taylor deformation index. Simulations of hundreds of three-dimensional deformable particles are presented in unbounded shear at concentrations up to 40%. Results include suspensions of solid elastic spheres, spherical capsules with elastic membranes, and model red blood cells.

5:06PM LT.00008 The Role of Deformation in the Rheology and Orientaiton of Noncolloidal Fiber Suspension , CYRUS AIDUN, JINGSHU WU, Georgia Institute of Technology — The microstructure and rheology of noncolloidal fibers suspended in Newtonian fluid are investigated with direct numerical simulation based on the external boundary force lattice-Boltzmann method. In this method, the flow is computed on a fixed regular "lattice", where each solid particle, or fiber in this case, is mapped onto a Lagrangian frame moving continuously through the domain. The motion and orientation of the fiber are obtained from Newtonian dynamics equations. The fiber bending ratio (BR) has significant influence on the flow rheology in the range BR $<$ 3, where for BR $>$ 3, the fiber can be considered as rigid. We show that in fiber suspension under shear, the relative viscosity decreases significantly as BR increases in the range of volume fraction from 1.7 to 12.4 percent considered in this study. This variation in relative viscosity appears to be well-correlated with the mean "contact" number. The effect of BR on the relative viscosity can also be explained based on the fiber orientation distribution. For increasing BR (more rigid fiber), the orientation distribution becomes narrower showing fibers oriented in the vicinity of the shear plane for a longer time, thus reducing the suspension shear viscosity. A small asymmetry of the fiber distribution is observed in the small BR regime, indicating the importance of fiber–fiber interaction and fiber deformation at BR$<$3.

5:19PM LT.00009 Rheology of a suspension of non-linear elastic particles in a viscous shear flow , HOWARD HU, TONG GAO, PEDRO PONTE CASTANEDA, University of Pennsylvania — The rheology of a suspension of two-dimensional (2D) non-linear elastic particles in a Newtonian viscous shear flow is studied. The particles are assumed to be neutrally buoyant and composed of neo-Hookean material.The deformation of the particle is governed by the Reynolds number (Re) and the Capillary number (Ca). In the Stokes flow regime, the particle deforms into a steady elliptic shape while material points inside experiencer a tank-treading like motion. In the dilute limit, a perturbation analysis is performed for a slightly deformed particle in a infinite media to calculate the effective properties of the mixture. 2D Einstein viscosity for rigid particle is recovered at the leading order. For particles with large deformation, the effective stress components and viscosity are calculated by assist of a numerical simulation. A monolithic finite element solver which uses Arbitrary Lagrangian-Eulerian moving mesh technique is implemented to solve the velocity, pressure and stress in both fluid and solid phase simultaneously. The results are also compared with previous works on linear elastic or viscoelastic particles in a viscous liquid.

5:32PM LT.00010 ABSTRACT HAS BEEN MOVED TO PE.00009 –

Monday, November 23, 2009 3:35PM - 5:45PM –

Session LU Separated Flows I 200I

3:35PM LU.00001 Dependence of the unsteady behavior of shock wave boundary layer interactions on the Reynolds number and on the state of separation[1] , PIERRE DUPONT, LOUIS SOUVEREIN, JEAN-PAUL DUSSAUGE, IUSTI/CNRS/Univ. Aix-Marseille, SUPERSONIC GROUP TEAM — The mean spatial organization and the unsteady behavior of a plane shock wave impinging on a turbulent boundary layer is investigated experimentally. Several parameters are considered: flow deviation, Mach and Reynolds numbers. Particular attention is paid to the incipient separation cases obtained at two Reynolds numbers with an order of magnitude of difference, obtained in different wind tunnels. The measurement technique is two component planar particle image velocimetry. The low frequency unsteadiness of the reflected shock is deduced by means of conditional statistics based on the existence of reverse flow. Upstream as well as downstream influences are considered. The results indicate that common mechanisms with the separated cases exist in the incipient cases, but with more or less importance, depending on the probability to observe reverse flow in the interaction. The effect of the Reynolds number will be shown to be negligible.

[1]UMR CNRS/Univ. Provence 6595

3:48PM LU.00002 Compressible turbulence properties in a shock wave boundary layer interaction[1] , JEAN-PAUL DUSSAUGE, SEBASTIEN PIPONNIAU, PIERRE DUPONT, IUSTI/CNRS/Univ. Aix-Marseille, SUPERSONIC GROUP TEAM — Shock induced separations are known to develop unsteadiness. Recently, a simple model was proposed to relate this unsteadiness to the dynamics of the mixing layer formed downstream of the separation shock. Such layers are known to be sensitive to compressibility. We will present turbulence measurements in the case of a Mach 2.3 shock reflection, performed by 2-component PIV. Turbulent data accuracy is tested in the upstream boundary layer, where no significant compressibility effects are expected on velocity fluctuations. The same turbulent data are shown in the interaction, where compressibility effects are likely. Results will be compared with data obtained in plane compressible mixing layers and in subsonic separated flows. It will be shown that, despite curvature effects and proximity of the wall, very similar behaviors are observed in the case of shock reflection. The main effect of compressibility is found to modify turbulence anisotropy, through a drastic reduction of vertical fluctuations, as in compressible mixing layers.

[1]UMR CNRS-Univ. Provence 6595

4:01PM LU.00003 Spanwise modulations in shock-induced boundary layer separations[1] , LIONEL LARCHEVEQUE, LIONEL AGOSTINI, JEAN-PAUL DUSSAUGE, IUSTI, UMR CNRS 6595, Provence University, Marseille, France, SUPERSONIC GROUP TEAM — Large-eddy simulations are carried out in order to clarify the origin of the large-scale spanwise modulations found experimentally within the separation bubble induced by a shock impinging on a turbulent boundary layer. Structures of various spanwise lengthscales are highlighted by means of short time averaged flowfield and spatial Fourier transforms. The individual influences of each of these modulations on the global dynamics of the flow are investigated by varying the spanwise extent of the computational domain. The effects of the spanwise boundary condition, ranging from periodicity to no-slip, are also investigated. Joint time-space spectral analyses are performed to quantify the timescale associated with each of the spanwise lengthscales. These timescales are compared with the timescales of the phenomena found in two-dimensional shock-boundary layer interaction with separation such as the shock motion, the mixing layer developing over the separated bubble and the turbulent structures stemming from the incoming boundary layer. The results are used to draw some conclusions on the possible origins of the spanwise modulations.

[1]Work partly supported by the CNES-ONERA ATAC programme.

4:14PM LU.00004 The leading-edge vortex and quasi-steady vortex shedding on an accelerating plate[1] , KEVIN CHEN[2], TIM COLONIUS, California Institute of Technology, KUNIHIKO TAIRA, Princeton University — A computational inquiry focuses on leading-edge vortex (LEV) growth and shedding during acceleration of a 2-D flat plate at a 30° angle of attack and at low Reynolds number. The plate accelerates from rest with a velocity given by a power of time ranging from 0 to 5. Comparison with Wagner's inviscid theory of airfoils reveals a lift coefficient peak across all powers during the initial LEV growth. The peak universally lasts four to five chord lengths of translation. A pattern of leading- and trailing-edge vortex shedding follows the shedding of the initial LEV. Plotted against Reynolds number, the nondimensional frequency, lift coefficient maxima, and lift coefficient minima of the shedding pattern closely match their values in the absence of acceleration. This leads to the support of a quasi-steady theory of vortex shedding, where dynamics are determined primarily by Reynolds number and not acceleration. Finally, the time scale of the lift augmentation matches well with the half-stroke of a flying insect, and supports a nondimensional vortex formation time close to 4.

[1]Partially supported by AFOSR FA9550-05-1-0369
[2]Present address: Princeton University

4:27PM LU.00005 An Experimental Investigation of Compressible Dynamic Stall on a Pitching Airfoil[1] , KATIE THORNE, PATRICK BOWLES, University of Notre Dame — A new facility has been designed and constructed at the University of Notre Dame to investigate dynamic stall on a 2-D pitching airfoil at high subsonic Mach numbers. This work is motivated by the need to investigate dynamic stall at conditions relevant to military helicopters. One focus of the experiments is to characterize the role of shock/boundary layer interactions during the pitching cycle. The new dynamic stall facility is integrated into a closed-loop, low turbulence wind tunnel capable of achieving test section Mach numbers in excess of M = 0.6. The design of the dynamic stall test section was focused on achieving reduced pitching frequencies of up to k = 0.2 and chord Reynolds numbers up to 5×10^6. The facility has the unique ability to execute non-harmonic pitching motions through the use of an actuated pitch link mechanism. Optical access is provided to allow the use of high-speed and Schlieren imaging. Thirty-one flush mounted Kulite dynamic pressure transducers provide the instantaneous unsteady surface pressure distribution over the airfoil. Initial dynamic stall measurements obtained in the new facility will be described.

[1]Project funded by Bell Helicopter.

4:40PM LU.00006 Measurements of Unsteady Turbulent Boundary Layer Separation under Conditions Relevant to Helicopter Rotors[1] , DAVID SCHATZMAN, FLINT THOMAS, University of Notre Dame — A unique experimental facility was developed to study unsteady turbulent boundary layer separation under conditions relevant to helicopter rotors. The facility provides the capability for unsteady turbulent boundary layer separation measurements of high spatial and temporal resolution. Leading edge plasma flow control on a stalled airfoil is used as a tool to impose an unsteady pressure gradient on turbulent boundary layer flow over a convex ramp section. 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. Phase locked PIV measurements are utilized to capture the dynamics of the unsteady turbulent boundary layer separation. High speed digital imaging of smoke flow visualization and simultaneous unsteady wall pressure records are used to track events that occur in the outer part of the boundary layer and propagate toward the wall. Joint hot-wire and unsteady wall pressure measurements are used to quantify these events during the unsteady separation process.

[1]Supported by ARO W911NF-07-0122.

4:53PM LU.00007 The Effects of Geometry on Trailing Edge Flows , SCOTT MORRIS, DAVID STEPHENS, University of Notre Dame — The flow field in the near wake of an airfoil is strongly influenced by the shape of the trailing edge. The shape will have and effect on the mean lift and drag, as well as the unsteady surface pressure that can lead to undesired aeroelastic and aeroacoustic phenomena. This talk will describe recent results that include unsteady wall pressure, PIV measurements of the velocity field, and radiated sound measurements from 9 different edge geometries with varied bluntness. The results will focus on the effects of the approach boundary layers on the separation point and the overall unsteadiness of the wake. The boundary conditions that lead to large scale vortex shedding will also be examined.

5:06PM LU.00008 Role of the recirculation region in the transition of a laminar separation bubble , O RAMESH, SOURABH DIWAN, Indian Institute of Science — The role of the recirculating region in the transition of a laminar separation bubble is explored in this work. The primary instability of a separation is convective and this inflexional mode is active along the mean inflection point locus located outside the mean dividing streamline. The region inside the dividing streamline and close to the wall – called the wall mode- however is seen to display a small region of negative production of turbulent kinetic energy. This has an interesting stabilizing role on the bubble dynamics. The negative production region is seen to increase in extent with increased values of maximum reversed flow velocity of the bubble. This is shown by doing a linear stability analysis of a mean velocity distribution of a model separation bubble. This negative production region is shown to have two important implications for the separation bubble: (a) The upper branch dynamics of the neutral stability curve is significantly affected by the negative production (b) while there is onset of absolute instability in the bubble with increased values of maximum reversed flow, it is seen that relatively higher values of reversed flow required for this onset are necessitated by the presence of the negative production region.

5:19PM LU.00009 Study on three-dimensional separation in diffusers[1] , ELBERT JEYAPAUL[2], PAUL DURBIN[3], Iowa State University — A family of three-dimensional diffusers having the same adverse pressure gradient but varying inlet aspect ratios have been studied by detached eddy simulation and Reynolds averaged computation. They provide a geometry for fundamental study of three-dimensional, turbulent separation. The diffuser has a rectangular cross section with two flared walls. RANS computations used the the standard $k-\omega$ and SST turbulence models. Experimental results of Cherry et al provide a reference against which the DES was validated. The SST model has been observed to be overly sensitive to inlet channel aspect ratio. Separation is observed to switch from top to the side wall at higher expansion ratios, while DES predicts separation to stay on the top wall. The recirculating region predicted by the model changes sides even for very small side angles. In a symmetrically–sloped diffuser the model does not predict a symmetric separation at higher side angles. The Reynolds stress models predict symmetric separation, so some of the shortcomings to the SST formulation emerge from the eddy viscosity assumption.

[1]NASA cooperative agreement NNX07AB29A
[2]Graduate student
[3]Professor

5:32PM LU.00010 Numerical Study of Flow past a Square Cylinder with an Angle of Incidence[1] , DONG-HYEOG YOON, KYUNG-SOO YANG, CHOON-BUM CHOI, Inha University, Korea — A parametric study has been carried out to elucidate the characteristics of flow past a square cylinder inclined with respect to the main flow for $Re \leq 300$. Reynolds number and angle of incidence (θ) are the key parameters which determine the flow characteristics. There exist two kinds of critical Reynolds numbers; flow becomes unsteady at a lower critical Reynolds number (Re_{c1}) and the two-dimensional time-periodic wake becomes unstable to three-dimensional disturbances at an upper critical Reynolds number (Re_{c2}). We present the two critical Reynolds numbers as a function of θ by using the Stuart-Landau equation for a steady flow and a fully-resolved Floquet stability calculation for a time-periodic wake, respectively. In particular, there are two different instability modes for a time-periodic wake, namely, one (mode A) associated with a long spanwise wavelength and the other (mode B) with a short spanwise wavelength. The spanwise wave numbers of the most unstable (or least stable) wave for the critical case are presented for each θ considered. We also report flow-induced forces on the cylinder, and flow patterns past the cylinder for the range of Re considered in this investigation, and attempt to present physical explanations about them.

[1]This work was supported by UVRC, Korea.

Monday, November 23, 2009 3:35PM - 5:45PM –
Session LV Propulsion Interactions 205A-D

3:35PM LV.00001 Interaction between the fore- and hind-wings in hovering flight of modelled dragonfly[1] , JIHOON KWEON, HAECHEON CHOI, Seoul National University — In the present study, we investigate the interaction between the fore- and hind-wings in hovering flight of modelled dragonfly using 3D numerical simulation. The three-dimensional wing shape is based on that of *Aeschna juncea* (Norberg 1972) and numerically realized using an immersed boundary method (Kim et al. 2001). The wing flapping motion is modelled using a sinusoidal function and the stroke plane angle is 60°. We consider 12 different phase differences between the fore- and hind-wings ($\phi = 0^\circ \sim 330^\circ$). The Reynolds number is 1,000 based on the maximum translational velocity and mean chord length. In counter stroke ($\phi = 180^\circ$), the wing-tip vortices from both wings are connected, generating an entangled wing-tip vortex (e-WTV). A strong downward motion induced by this vortex decreases the vertical force in the following stroke (Kweon & Choi 2008). In parallel stroke ($\phi = 0^\circ$), both wings meet e-WTV during the upstroke and thus the decrease of vertical force is small. At $\phi = 270^\circ$, although e-WTV is generated on a relatively narrow region, the hind-wing moves downward along with e-WTV, resulting in a significant reduction of vertical force on the hind-wing. Therefore, the sum of vertical forces on both wings is maximum with parallel stroke and minimum at $\phi = 270^\circ$. The power required has a similar trend to the vertical force and thus the efficiency does not show a large variation with the phase difference.

[1]Supported by the NRL Program.

3:48PM LV.00002 Wing-Wake Interactions between Ipsilateral Wings in Dragonfly Flight , HAIBO DONG, ZONGXIAN LIANG, Wright State University — Bilateral and ipsilateral wing-wing interactions can be commonly observed in insect flights. As a representative example of ipsilateral wing-wing interaction, dragonflies in flight have been widely studied. An important fact is that the flow over their hindwings is affected by the presence of the forewings. Wake capture and phase-change play very important role on aerodynamic performance of the hindwings We present a direct numerical simulation of a modeled dragonfly (Aeshna juncea) in slow flight as studied in Azuma et al (JEB 1985). Realistic morphologies of wing, body, and kinematics are used for maximum including wing and body features of a dragonfly. This work aims to study the relations between wake-topology and aerodynamic performance due to wing-wing and wing-wake interactions of dragonfly ipsilateral wings. DNS results are also compared with Local Momentum Theory (Azuma et al).

4:01PM LV.00003 Unsteady aerodynamics of dragonfly using a wing-wing model from the perspective of a force decomposition[1] , CHIN-CHOU CHU, National Taiwan University, CHIEN C. CHANG, National Taiwan University and Research Center for Applied Sciences, Academia Sinica, CHEN-TA HSIEH, National Taiwan University — The lift and thrust associated with insect flight strongly depend on the complex wake patterns produced by wing-wing and wing-wake interactions. We propose to investigate the aerodynamics of dragonfly using a simplified wing-wing model from the perspective of many-body force decomposition (JFM 600, p95) and the associated force elements. The aerodynamic force, lift or thrust, of the wing-wing system is analyzed in terms of its four constituent components, each of which is directly related to a physical effect. These force components for each individual wing include two potential contributions credited to the wing motion itself, contribution from the vorticity within the flow, and contributions from the surface vorticity on its and other wing's surfaces. The potential contribution due to added-mass effect is often non-negligible. Nevertheless, the major contribution to the forces comes from the vorticity within the flow. The relative importance of these components relies heavily on the motions of the two wings such as the respective angles of attack, the amplitude and speed of translational motions, and the amplitude and speed of wing rotations. In addition to the dynamic stall vortex, several important mechanisms of high lift or thrust are also identified.

[1]Partically supported by NSC under No's NSC97-2221-E-002-223-MY3 and NSC97-2221-E-002-048.

4:14PM LV.00004 The interaction of two bodies falling in tandem , NICOLAS BROSSE, PATRICIA ERN, Institut de Mecanique des Fluides de Toulouse, France — We have investigated experimentally the interaction of two identical bodies falling in a fluid at rest at intermediate Reynolds numbers ($100 < Re < 300$). The bodies are disks of various diameter-to-thickness ratios ($2 < d/t < 10$) and of density close to the fluid one. They are released either consecutively or simultaneously at two different locations. The path of the bodies when they fall separately is either rectilinear or a periodic zigzag depending on the corresponding values of d/t and Re. We will focus here on the case of two bodies released consecutively and exhibiting a rectilinear path when they fall separately. The motion of the bodies was recorded by two travelling cameras. While the forebody (the first body released) follows a rectilinear path, the aftbody (the second body released) accelerates thanks to the forebody's wake and oscillates when Re is close to the critical value of appearance of the zigzag motion. Though the forebody also accelerates, the aftbody eventually catches up the forebody. Afterwards, thick bodies ($d/t = 3$) separate, whereas thinner bodies ($d/t > 5$) continue their fall together. We will describe quantitatively the characteristics of the motion of the two bodies before and after grouping.

4:27PM LV.00005 Hydrodynamic interactions between flagella , PIETER JANSSEN, MICHAEL GRAHAM, UW-Madison — Many bacteria, such as *E. coli*, use several rotating flagella to propel themselves at low-Reynolds numbers. If the flagella are all rotating counter-clockwise, they bundle up, and the cell moves at great speed. However, if one flagellum starts to rotate clockwise, it disentangles from the bundle, and the cell starts to rotate randomly. After a while, the rotation of all flagella becomes counter-clockwise again, and the cell starts moving again, now in a different direction. The bundling and disentangling is poorly understood from a fluid mechanics point of view. We investigate the hydrodynamic interactions between flagella that may lead to the bundling. Flagella are modeled as series of spheres connected through hinges with bending and twisting resistance. Hydrodynamic interaction between the spheres is incorporated through standard expressions. The cell body is described with a boundary-integral method. Synchronization between the flagella is shown, and we investigate the effect of stiffness, pitch and length of the flagella, and of the hook connecting the flagellum to the cell. Furthermore, we show the effect on the orientation, rotation and speed of the cell body under the influence of multiple flagella.

4:40PM LV.00006 Synchronization of Swimming Microorganisms , GWYNN ELFRING, ERIC LAUGA, Department of Mechanical and Aerospace Engineering, University of California San Diego — Flagellated eukaryotic cells (such as spermatozoa) have been observed to synchronize their flagella when swimming in close proximity. Using a 2D model, we find that hydrodynamic interactions alone can lead to synchronization if the waveforms of the flagella display front-back asymmetry. Depending on the nature of the asymmetry, the phase-locked conformation can minimize or maximize the energy dissipated by the co-swimming cells. We show that due to kinematic reversibility, this front-back asymmetry is necessary for synchronization in a Newtonian fluid, and discuss the differences in a non-Newtonian fluid.

4:53PM LV.00007 Sychronization of flagella and cilia due to viscous interactions[1] , DAVID GAGNON, BIAN QIAN, HONGYUAN JIANG, THOMAS POWERS, KENNETH BREUER, Brown University — Motivated by the observed coordination of nearby beating cilia and rotating bacterial flagella, we use a scaled model experiment to show that hydrodynamic interactions can cause synchronization between rotating paddles driven at constant torque in a very viscous fluid. Systems with two and three paddles are explored, and interactions between symmetric and asymmetric paddles are tested. For two-paddle systems, synchronization is only observed when the shafts supporting the paddles have some flexibility, and the phase difference in the synchronized state depends on the symmetry of the paddles. Calculations using the method of regularized stokeslets and simple analytic theory match the experimental observations well.

[1]Supported by NSF

5:06PM LV.00008 Wake-mediated synchronization and drafting in coupled flags , SILAS ALBEN, Georgia Tech — A recent experiment has shown "inverted drafting" in flags: the drag force on one flag is increased by excitation from the wake of another. Here we use vortex sheet simulations to show that inverted drafting occurs when the flag wakes add coherently to form strong vortices. By contrast, normal drafting occurs for higher-frequency oscillations, when the vortex wake becomes more complex and mixed on the scale of the flag. The types of drafting and dynamics (synchronization and erratic flapping) depend on the separation distance between the flags. For both tandem and side-by-side flags in synchronized flapping, the phase difference depends nearly monotonically on separation distance. These results provide a framework for how bodies interact through their wakes, and may be used to identify optimal rigidities and separation distances for bodies in collective locomotion.

5:19PM LV.00009 Fish schooling as a basis for wind farm design[1] , ROBERT WHITTLESEY, JOHN DABIRI, California Institute of Technology — It is known that horizontal axis wind turbines (HAWT) suffer from reduced aerodynamic efficiency when in close proximity to neighboring turbines. In contrast, recent work has shown that closely spaced vertical axis wind turbines (VAWT) may benefit from enhanced performance, reducing the associated land use for VAWT wind farm installations. A potential flow model of VAWT interactions is developed to determine configurations that optimize the power output of the array. A geometric arrangement based on fish schooling has been shown to significantly increase the array performance as measured by an Array Power Coefficient, which compares the average performance of turbines in the array to an isolated turbine. The results suggest that significant gains may be obtained through careful arrangement of VAWTs, showing up to a two order of magnitude decrease in land use (equivalently, a two order of magnitude increase in power density) compared to HAWTs.

[1]This work is supported by the NSF Program in Energy for Sustainability.

5:32PM LV.00010 Role of Flexibility in Thrust Production of a Mechanical Swimming Lamprey , MEGAN LEFTWICH, ALEXANDER SMITS, Princeton University — To develop a comprehensive model of lamprey locomotion, we use a robotic lamprey as a means of investigating the wake structure during swimming with an anatomically designed tail of varying degrees of flexibility. 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 behind the most flexible tail show a 2P wake structure that quickly looses coherence as it is convected downstream. This is in contrast to the strongly coherent and symmetrical 2P wake seen in previous experiments using a rigid, rectangular tail. The project is supported by NIH CNRS Grant 1R01NS054271.

Monday, November 23, 2009 3:35PM - 5:45PM –

Session LW Biofluids VIII: Cellular II 208A-D

3:35PM LW.00001 Mechanics of bacterial biofilm expansion: differentiation and nutrient deprivation , AGNESE SEMINARA, TOMMY ANGELINI, School of Engineering and Applied Sciences, Harvard University, ROBERTO KOLTER, Harvard Medical School, Harvard University, DAVID WEITZ, MICHAEL BRENNER, School of Engineering and Applied Sciences, Harvard University — Biofilms are sessile colonies of micro-organisms, usually encased in a protective extracellular matrix and associated to a surface. Cells inside a biofilm differentiate in precise patterns in space and time, so that biofilms are often compared to multicellular organisms. Nutrient deprivation is thought to be the main trigger for the onset of differentiation. Here we explore this idea quantitatively by modeling the mechanics of biofilm growth coupled to reaction and diffusion of nutrient concentration. The results give insight on the time evolution of biofilm morphology and provides a quantitative description of the stresses developed within the biofilm during expansion. We develop a theoretical framework applicable to a variety of microorganisms and compare the predictions to time-lapse microscopy data on Bacillus Subtilis biofilms.

3:48PM LW.00002 Shear-induced adhesion of bacterial cells[1] , SIGOLENE LECUYER, ROBERTO RUSCONI, YI SHEN, ALISON FORSYTH, Harvard University, HOWARD STONE, Princeton University — Bacterial adhesion is the first step in the development of surface-associated communities known as biofilms. The formation of these microbial structures is the cause of many different problems in medical devices and industrial water systems. Despite an extensive literature, the underlying mechanisms of the initial reversible attachment are not fully understood. We have investigated the effects of hydrodynamics on the probability of adsorption and detachment of bacteria on model surfaces by using phase-contrast microscopy in straight microchannels. In this way we have been able to measure the time that each bacterium spends on the surface and to analyze the mobility as a function of the flow rate. The main finding of our experiments and analyses is a counter-intuitive enhanced adhesion as the shear stress is increased over a wide range of shear rates.

[1]This work was supported by BASF via the BASF Advanced Research Initiative at Harvard University.

4:01PM LW.00003 Leukocyte transport by red blood cells in a microvessel , JONATHAN FREUND, University of Illinois at Urbana-Champaign — A simulation model is used to study the transport of relatively large, spherical, and stiff white blood cells (leukocytes) by the relatively smaller and highly flexible red cell as they flow in the microcirculation. Their interaction dynamics are thought to be an important component of the inflammation response, in which leukocytes bind to the walls of blood vessels. The red cells are modeled in the simulations as highly deformable three-dimensional shells encasing a Newtonian fluid, and the viscous-flow equation is solved via a boundary integral formulation in which the cell shapes discretized by global spectral basis functions. For slow flow rates, it is found that the leukocyte is predominantly adjacent the vessel walls, whereas for faster flow rates this configuration appears to become unstable and the leukocyte traverses the whole vessel in a seemingly random fashion. For the straight round tubes simulated thus far, the stable leukocyte stand-off distance is always beyond the range of the binding molecules that capture it, which suggests that vessel inhomogeneities or interactions with other white cells are needed to create contact and thereby binding with the vessel walls.

4:14PM LW.00004 Role of thermal fluctuations in vesicle dynamics , KONSTANTIN TURITSYN, Los Alamos National Laboratory, SERGEY VERGELES, VLADIMIR LEBEDEV, Landau Institute for Theoretical Physics — Lipid bilayer vesicles can exhibit several regimes of motion when subjected to external flow: tumbling, tank-treading, trembling. Theoretical predictions based on deterministic models proved to be very successful in describing the corresponding phase diagram on qualitative level. However, recent experimental studies [Deschamps et.al., Phys.Rev.Lett. 178102 (2007)] identified significant quantitative discrepancies between theory and experiment, related mainly to the transition between tumbling and trembling regimes. Here we show that some of these discrepancies can be attributed to the role of thermal fluctuations. We extend the theoretical model to account for their effect. Fluctuations of the membrane give rise to effective compressibility, and lead to renormalization of its surface area. Resulting dynamical equations are still simple enough to be studied analytically and the resulting phase diagram is consistent with new experimental observations.

4:27PM LW.00005 Modeling of Fluid-Membrane Interaction in Cellular Microinjection Process[1] , MEHDI KARZAR-JEDDI, JHON DIAZ, NEJAT OLGAC, TAI-HSI FAN, University of Connecticut — Cellular microinjection is a well-accepted method to deliver matters such as sperm, nucleus, or macromolecules into biological cells. To improve the success rate of *in vitro* fertilization and to establish the ideal operating conditions for a novel computer controlled rotationally oscillating intracytoplasmic sperm injection (ICSI) technology, we investigate the fluid-membrane interactions in the ICSI procedure. The procedure consists of anchoring the oocyte (a developing egg) using a holding pipette, penetrating oocyte's zona pellucida (the outer membrane) and the oolemma (the plasma or inner membrane) using an injection micropipette, and finally to deliver sperm into the oocyte for fertilization. To predict the large deformation of the oocyte membranes up to the piercing of the oolemma and the motion of fluids across both membranes, the dynamic fluid-pipette-membrane interactions are formulated by the coupled Stokes' equations and the continuum membrane model based on Helfrich's energy theory. A boundary integral model is developed to simulate the transient membrane deformation and the local membrane stress induced by the longitudinal motion of the injection pipette. The model captures the essential features of the membranes shown on optical images of ICSI experiments, and is capable of suggesting the optimal deformation level of the oolemma to start the rotational oscillations for piercing into the oolemma.

[1]Supported by National Science Foundation grant CBET-0828733.

4:40PM LW.00006 The role of ion electrophoresis in electroporation-mediated molecular delivery , JIANBO LI, HAO LIN, Rutgers University — Electroporation is a widely applied technique to deliver active molecules into the cellular compartment, to perform a variety of tasks such as gene therapy and directed stem cell differentiation. In this technique, an electric field transiently permeabilizes the cellular membrane to facilitate molecular exchange. While the permeabilization process is relatively well-understood, the transport mechanisms for molecular delivery are still under debate. In this work, the role of ion electrophoresis in electroporation-mediated molecular delivery is investigated using numerical simulations. The result indicates that ion electrophoresis is the dominant mode of transport in the delivery of small charged molecules. Furthermore, the achievable intracellular concentration is strongly influenced by the conductivity difference between the cytoplasm and the buffer, a phenomenon known as "field-amplified sample stacking". The result agrees well with the fluorescence measurement by Gabriel and Teissié (1999), and suggests a new possibility to simultaneously improve cell viability and efficiency in electroporation-mediated molecular delivery.

4:53PM LW.00007 Stability of a Lipid Bilayer Membrane Subjected to a DC Electric Pulse , JONATHAN SCHWALBE, Northwestern University, PETIA VLAHOVSKA, Dartmouth College, MICHAEL MIKSIS, Northwestern University — An analytical theory is developed to study the dynamics of a lipid bilayer membrane subjected to a DC electric pulse. The thin lipid membrane is impermeable to ions and thus acts as a capacitor. The model consists of conservation of current, which obeys Ohm's law, and the Stokes equations to describe fluid motion. The effects of membrane fluidity and incompressibility, variations in lipid density along the monolayers, and resistance to bending are taken into account. Small amplitude perturbations of a planar membrane are considered. The result is a time dependent system of equations for the growth rate as a functions of wave number. Variation of the applied voltage and a difference in the conductivities of the bulk fluids yield a long-wave instability in the system. The theory highlights that the membrane charging time is critical for the instability. Our theoretical findings are relevant to understanding the physical mechanisms of electroporation of biomembranes.

5:06PM LW.00008 Soluble surfactants favorably modify fluid structure and wall shear stress profiles during near-occluding bubble motion in a computational model of intravascular gas embolism[1] , T.N. SWAMINATHAN, P.S. AYYASWAMY, D.M. ECKMANN, University of Pennsylvania — Finite sized gas bubble motion in a blood vessel causes temporal and spatial gradients of shear stress at the endothelial cell surface lining the vessel wall as the bubble approaches the cell, moves over it and passes it by. Rapid reversals occur in the sign of the shear stress imparted to the cell surface during this motion. The sign-reversing shear is a potently coupled source of cell surface mechanical stretch, potentiating cell injury. The presence of a suitable soluble surfactant in the bulk medium considerably reduces the level of the shear stress gradients imparted to the cell surface as compared to an equivalent surfactant-free system. The bubble shape and the film thickness between the bubble and the vessel wall are also different. Furthermore, the bubble residence time near the proximity of a cell surface changes in comparison. These results based on our modeling may help explain several phenomena observed in experimental studies related to gas embolism, a significant problem in cardiac surgery and decompression sickness.

[1]Supported by: ONR-N00014-08-1-0436, NIH-R01-HL067986, NASA-NNX07AP97G.

5:19PM LW.00009 Three-dimensional simulation of 10^3 deformable capsules in Poiseuille flow[1] , R. MURTHY KALLURI, SAI DODDI, PROSENJIT BAGCHI, Rutgers University — Three-dimensional simulation using front-tracking methods are presented on the motion of 10^3 deformable elastic capsules at semi-dense suspension in Poiseuille flow in microvessels, typical of microcirculation and microfluidic devices. The computational framework considered here can resolve the dynamics of individual deformable cell with high fidelity, yet can consider a large number of hydrodynamically interacting cells. In the simulations, the flow field is resolved using up to 300^3 Eulerian grid points, and each capsule surface is resolved by up to 1280 triangular elements. Flow visualization, and analysis of cell trajectory and velocity of the multi-file motion are presented as functions of the cell deformability, and volume fraction. The simulations are computation- and data- intensive, and the first of their kind in the context of deformable capsule suspension. They provide a wealth of information on the dynamics of semi-dense suspension of liquid capsules, in particular, and of deformable particles, in general. The numerical results allow us to analyze various microrheological phenomena, such as the particle migration and formation of near-wall depletion layer, plug-flow velocity, and Fahraeus and Fahraeus-Lindqvist effects.

[1]Funded by NSF (BES-0603035, CTS-0625936).

5:32PM LW.00010 Manipulation of red blood cells with electric field[1] , HOSSAIN SABOONCHI, ASGHAR ESMAEELI, Southern Illinois University at Carbondale — Manipulation of bioparticles and macromolecules is the central task in many biological and biotechnological processes. The current methods for physical manipulation takes advantage of different forces such as acoustic, centrifugal, magnetic, electromagnetic, and electric forces, as well as using optical tweezers or filtration. Among all these methods, however, the electrical forces are particularly attractive because of their favorable scale up with the system size which makes them well-suited for miniaturization. Currently the electric field is used for transportation, poration, fusion, rotation, and separation of biological cells. The aim of the current research is to gain fundamental understanding of the effect of electric field on the human red blood cells (RBCs) using direct numerical simulation. A front tracking/finite difference technique is used to solve the fluid flow and electric field equations, where the fluid in the cell and the blood (plasma) is modeled as Newtonian and incompressible, and the interface separating the two is treated as an elastic membrane. The behavior of RBCs is investigated as a function of the controlling parameters of the problem such as the strength of the electric field.

[1]NSF

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MA Turbulent Boundary Layers: Experiments III 101A

8:00AM MA.00001 Scaling of near-wall turbulence in pipe flow[1] , MARCUS HULTMARK, SEAN BAILEY, ALEXANDER SMITS, Princeton University — Experiments were conducted in the Princeton/ONR Superpipe. Profiles of the streamwise velocity component were measured in fully developed pipe flow at Reynolds numbers from 25×10^3 to 150×10^3. The turbulence intensity profiles non-dimensionalized with inner coordinates indicate that the magnitude of the near-wall peak is invariant with Reynolds number in both location and magnitude. The results agree with previous pipe flow data that have sufficient spatial resolution to avoid spatial filtering effects, but stand in contrast to similar results obtained in boundary layers, where the peak displays a strong Reynolds number dependence, although its position, in wall units, is fixed at the same location as in pipe. Although it is expected that the current experiments have sufficient spatial resolution at all Reynolds numbers, experiments are being conducted at constant l^+ to investigate the potential impact of probe spatial filtering on these results.

[1]Supported under NSF Grant CTS-0625268, monitored by Bill Schultz, and ONR Grant N00014-09-1-0263, monitored by Ron Joslin.

8:13AM MA.00002 Reynolds number effects on scale energy analysis of turbulent boundary layers[1] , NEELAKANTAN SAIKRISHNAN, ELLEN LONGMIRE, University of Minnesota Twin Cities, IVAN MARUSIC, University of Melbourne — Scale energy analysis combines two approaches of studying wall- bounded turbulent flows - analysis in physical space and analysis in scale space. Previously, scale energy analysis has been performed on DNS channel flow data for a range of friction Reynolds numbers $\mathrm{Re}_\tau = 180 - 934$ and dual plane PIV boundary layer data at $\mathrm{Re}_\tau = 1100$. The dual plane technique allows determination of the full velocity gradient tensor in the measurement plane. Dual Plane PIV data were acquired in streamwise-spanwise planes in the logarithmic region of a water channel boundary layer at two higher Reynolds numbers - $\mathrm{Re}_\tau = 2400$ and 3000. The results of this study will be described and compared with the lower Re data. It is observed that in general, the production and scale transfer terms of the turbulent kinetic energy increase with increasing Reynolds number. The cross-over scale, which divides the range of scales into a transfer-dominated region and a production- dominated region, increases with increasing Reynolds numbers, resulting in a larger range of transfer-dominated scales at higher Reynolds numbers.

[1]Support from APS (CTS-0324898) and ARC.

8:26AM MA.00003 Vortex organization in wall turbulence: inner or outer scaling?[1] , MICHELE GUALA, JEFF LEHEW, GALCIT, California Institute of Technology, MEREDITH METZGER, Dep. Mechanical Engineering, University of Utah, BEVERLEY MCKEON, GALCIT, California Institute of Technology — Simultaneous hotwire measurements in the near-neutral atmospheric surface layer at $Re_\tau = \delta u_\tau/\nu \simeq 10^6$ are compared with time-resolved PIV measurements in a flat plate turbulent boundary layer at $Re_\tau = 5 \cdot 10^2$. We observe in both flows a similar strong signature in the two point correlation function of the streamwise velocity fluctuation near the wall, confirming the key statistical role of ramp-like vortex organization. However, at the lower Reynolds number, the organized structures were observed to extend up to the boundary layer thickness δ, implying that in the atmospheric surface layer we should observe similar patterns up to heights of the order of 50 m, providing outer scaling holds. The effect of the Reynolds number on the scaling of ramp-like structures, hairpins and hairpin packets (Adrian, 2007) is investigated.

[1]We acknowledge the support by ONR grant #N00014-08-1-0897 (program manager Ron Joslin).

8:39AM MA.00004 PIV Characterization of Turbulent Channel Flow with Rib Patterned Superhydrophobic Walls , JOSEPH PRINCE, BRADY WOOLFORD, DANIEL MAYNES, BRENT WEBB — We report PIV measurements characterizing turbulent flow in a channel with superhydrophobic surfaces, structured and wetting surfaces, and smooth bottom surfaces. The superhydrophobic and structured surfaces were fabricated with alternating ribs and cavities. Both longitudinal and transverse rib/cavity orientations were considered. The widths of the ribs and cavities were 8 and 32 micrometers, respectively, and the cavity depths were 15 micrometers. PIV measurements were acquired over the Reynolds number range of 4800 – 10000. The smooth bottom wall was used as a basis for comparison. The hydraulic diameter of the channel was nominally 8.2 mm with an aspect ratio of 8.9. A spanwise-averaged velocity profile was obtained at the channel centerline from the PIV data. The time-averaged velocity profiles reveal no discernible time-mean slip velocity at the superhydrophobic wall. However, the different surfaces affect the turbulence intensities, total and turbulent shear stress distributions, turbulence production in the channel, and local friction factors. Superhydrophobic surfaces with the ribs and cavities aligned with the flow show an 11% decrease in the friction factor while the same surfaces aligned in the transverse direction show a modest increase in the friction factor.

8:52AM MA.00005 Behavior of Different Turbulent Length Scales in a Boundary Layer with Isotropic Freestream Turbulence[1] , SHEILLA TORRES-NIEVES, JOSE LEBRON, BRIAN BRZEK, LUCIANO CASTILLO, Rensselaer Polytechnic Institute, HYUNG-SUK KANG, CHARLES MENEVEAU, The Johns Hopkins University, RAUL B. CAL, Portland State University — The effects of different length scales on interactions between nearly isotropic freestream turbulence (FST) and a favorable pressure gradient, turbulent boundary layer (TBL) over a rough surface are studied. Measurements are obtained using Laser Doppler and Hotwire Anemometry, at $Re_\theta \leq 4{,}300$. An active grid is used to generate FST levels of up to 7%. Profiles of mean turbulent statistics show that the classical view of TBL flows is not able to collapse the data when FST is present. The different effects of FST on the streamwise and wall-normal variances result in that the addition of isotropic FST promotes anisotropy in the boundary layer. Second-order structure functions are examined to identify which turbulence length scales contribute mostly to creating the anisotropy. The analysis demonstrates that the effect of FST resides in a wide range of length scales, and is not limited to the largest scales of the flow as in the zero pressure gradient case. Spectral analysis is performed to more accurately identify the relevant scales.

[1]Supported by NSF (GRF, HRD-0202171, CBET-0553314) and the Ford Foundation Diversity Fellowship.

9:05AM MA.00006 Experimental evidence of the Self Sustaining Process in a flat plate boundary layer , THOMAS DURIEZ, JEAN-LUC AIDER, JOSE EDUARDO WESFREID, PMMH / ESPCI — Streamwise velocity streaks and streamwise vorticity are both key features of the boundary layer and are involved into its transition to turbulence. It has been proposed that a Self Sustaining Process (SSP) exists between these structures from a given Reynolds number. We use bluff-body vortex generators to steadily force a flat plate boundary layer with counter-rotating streamwise vortices (CRSV) in a hydrodynamic tunnel at moderate Reynolds numbers. Using 2-Component 2-Dimensional Particle Image Velocimetry we reconstruct the 3-Dimensional time averaged flow field for both streamwise and spanwise components. By associating the modulation on the spanwise component to the CRSVs and the modulation on the streamwise component to the velocity streaks we witness the evolution of these structures. Using a parametric study on the Reynolds number we study the evolution of the streamwise streaks transient growth. Monitoring the evolution of the amplitude of the CRSV we show the transition from their decay at low Reynolds number to the existence of the SSP. Looking at the instantaneous vector fields we propose the destabilization of the streamwise streaks as the process responsible to the existence of a critical Reynolds number for the SSP.

9:18AM MA.00007 The Mechanism of Heat Transfer Augmentation in Stagnation Flow Subject to Freestream Turbulence , DAVID HUBBLE, Virginia Tech, TOM DILLER, PAVLOS VLACHOS — A physical model is presented which predicts the time-resolved heat transfer coefficient based on the properties of the coherent structures present. Water tunnel experiments have been performed to investigate the mechanism of heat transfer augmentation in stagnation flow subject to freestream turbulence. The experiment combined Time-Resolved Digital Particle Image Velocimetry with an array of simultaneous time-resolved heat transfer measurements. Passive grids produced freestream turbulence with an intensity of 5% at length scales of 1cm, 2cm, and 3cm. The measurements reveal flow fields dominated by coherent structures whose number and strength strongly correlate with the length scale of the freestream turbulence. By examining the transient circulation and location of the identified structures, we observe that stretching and vorticity amplification significantly affects the near-wall flow. The transient heat transfer correlates well with the flow field induced by these structures. The time-resolved model developed represents a great advance over previous time-average predictors.

9:31AM MA.00008 Scaling of Polymer Degradation Rate within a High-Reynolds-Number Turbulent Boundary Layer , BRIAN ELBING, MICHAEL SOLOMON, MARC PERLIN, DAVID DOWLING, STEVEN CECCIO, University of Michigan — An experiment conducted at the U.S. Navy's Large Cavitation Channel on a 12.9 m long flat-plate test model produced the first quantitative measurements of polymer molecular weight within a turbulent boundary layer. Testing was conducted at speeds to 20 m/s and downstream distance based Reynolds numbers to 220 million. These results showed that the rate of polymer degradation by scission of the polymer chains increases with increased speed, downstream distance and surface roughness. With the surface fully rough at 20 m/s there was no measureable level of drag reduction at the first measurement location (0.56 m downstream of injection). These results are scaled with the assumption that the rate of degradation is dependent on the polymer residence time in the flow and the local shear rate. A successful collapse of the data within the measurement uncertainty was achieved over a range of flow speed (6.6 to 20 m/s), surface roughness (smooth and fully rough) and downstream distance from injection (0.56 to 9.28 m).

9:44AM MA.00009 Superhydrophobic Surfaces for Turbulent Drag Reduction: Effect of Geometry and Reynolds Number , ROBERT DANIELLO, JONATHAN P. ROTHSTEIN, University of Massachusetts, Amherst — Recently, it has been demonstrated that superhydrophobic surfaces are capable of reducing drag in turbulent flows. Superhydrophobic surfaces are chemically hydrophobic with micron or nanometer scale surface features which can support a shear-free air-water interface. In this talk, we will consider the effect of microfeature geometry and flow velocity on the observed drag reduction. Microridge geometries from 15μm to 60μm will be considered with shear free area ratios from 20% to 80%. Drag reductions are found to increase with increasing shear free area ratio up to 75%. For any given shear free area ratio, drag reduction was seen to initiate at lower Reynolds numbers for larger microfeature spacings. Experiments were conducted over the range of Reynolds numbers $1000<Re<100000$, demonstrating that drag reduction in the turbulent regime is distinct from drag reduction observed over superhydrophobic surfaces in the laminar regime and persists well past transition to turbulence.

9:57AM MA.00010 Drag reduction in flows over superhydrophic surfaces[1] , CHARLES PEGUERO, Brown University, CHARLES HENOCH, NUWC, KENNETH BREUER, Brown University — Recent research has suggested that large reductions in both laminar and turbulent skin friction might be realized in water flows over over superhydrophobic surfaces due to the modification of the no-slip boundary condition at the liquid-solid interface by a thin layer of trapped air. In our work, superhydrophobic surfaces have been fabricated by first laser-etching a textured pattern into an aluminum baseplate, and then coating the surface with a hydrophobic coating. The superhydrophbic characteristics and the presence of a trapped air layer have been characterized at a variety of operating pressures. The drag characteristics of the baseline and superhydrophobic surfaces have also been measured under both laminar and turbulent flow conditions using three different measurement apparatus - a narrow gap laminar flow channel, a large gap turbulent flow channel and a closed return water tunnel. The results from these measurements will be presented and discussed in detail.

[1]Supported by ONR

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MB Turbulence Modelling III 101B

8:00AM MB.00001 Progress in Algebraic Reynolds Stress Model development for compressible flows , CARLOS GOMEZ, SHARATH GIRIMAJI, Texas A&M University — The standard algebraic Reynolds stress modeling (ARSM) approach is used to develop closures for compressible turbulence. A new rapid-pressure strain correlation model that is sensitive to gradient (Ma_g) and turbulent (Ma_t) Mach numbers is employed in the closure development. At low Ma_g, the pressure-strain correlation model assumes its standard incompressible form and the ARSM of Girimaji (1996) is recovered. Based on the analysis of the Navier-Stokes equation in the rapid distortion limit, it is suggested that at intermediate Ma_g the effects of pressure, on an average, is to counter that of inertia. This leads to a balance between production and pressure-strain correlation leading to no temporal change in Reynolds stresses in this regime. In the high Ma_g regime, the effect of pressure is negligible compared to inertia thus yielding a fully explicit algebraic Reynolds stress model. The complete proposed model is tested in a 2D supersonic mixing layer. Similarity profiles of mean streamwise velocity and turbulence quantities are compared with the experimental data of Goebel and Dutton (1991). Numerical mixing layer spreading rates are found to be in relatively good agreement with experimental data.

8:13AM MB.00002 Progress in the development of explicit Algebraic passive Scalar Flux Model (ASFM) for compressible flows , MONA KARIMI, SHARATH GIRIMAJI, Aerospace Engg., Texas A&M University, TURBULENCE RESEARCH GROUP TEAM — Closure modeling of turbulent scalar flux represents an important field of active research in turbulent combustion. Invoking the weak-equilibrium limit of turbulence, the evolution equation for the scalar flux is simplified. Utilizing Representation theory, the resulting algebraic equation is solved for normalized scalar flux. The novelty of the work lies in the extension of this procedure to compressible flows, employing the pressure-scalar gradient correlation that is valid for a wide range of Gradient Mach numbers: (i) the incompressible model of the pressure-scalar gradient is retained at low Mach numbers, (ii) the pressure effects are take to be negligible compared to inertial term at very large Mach numbers, and (iii) the action of pressure is assumed to nullify the inertial effects at intermediate Mach numbers. It is demonstrated that the eddy diffusivity depends, not just on the mean scalar gradient, but also on the anisotropy of the velocity field. The dependence of eddy diffusivity on strain-rate, rotation-rate, and time scale ratio at different Gradient Mach numbers is examined. Additionally, simulations employing the model will be presented.

8:26AM MB.00003 A Method for Interface-Turbulence Forcing in Hybrid LES/RANS Simulations , SUAD JAKIRLIC, BJOERN KNIESNER, Darmstadt University of Technology — A computational strategy coupling near-wall, eddy-viscosity-based RANS models with LES in a two-layer Hybrid LES/RANS (HLR) scheme is proposed in the present work. Key questions concerning the coupling of both methods, the inherently steady RANS method and highly-unsteady LES method, are closely connected to the treatment at the interface separating both sub-regions: (1) the exchange of the variables across the LES/RANS interface was adjusted by implicit imposition of the condition of equality of the modeled turbulent viscosities; (2) second issue is the utilisation of a self-adjusting interface position in the course of the simulation; (3) the third issue, the present work is focussing on, addresses the usage of a special forcing technique, which compensates the loss of information due to strong damping in the RANS region by creation of artificial and correlated fluctuations using a method originating from a digital-filter-based generation of inflow data for spatially developing DNS and LES due to Klein et al. (2003). Herewith, the recovery of the fluctuations on the LES side of the interface is accelerated. The model validation is conducted by computing numerous wall-bounded configurations of different geometrical complexity featuring different mean flow and turbulence phenomena.

8:39AM MB.00004 Towards a Dynamic DES model[1] , PRAMOD SUBBAREDDY, GRAHAM CANDLER, University of Minnesota — Hybrid RANS/LES methods are being increasingly used for turbulent flow simulations in complex geometries. Spalart's detached eddy simulation (DES) model is one of the more popular ones. We are interested in examining the behavior of the Spalart-Allmaras (S-A) Detached Eddy Simulation (DES) model in its "LES mode." The role of the near-wall functions present in the equations is analyzed and an explicit analogy between the S-A and a one-equation LES model based on the sub-grid kinetic energy is presented. A dynamic version of the S-A DES model is proposed based on this connection. Validation studies and results from DES and LES applications will be presented and the effect of the proposed modification will be discussed.

[1]This work was sponsored by the Air Force Office of Scientific Research under grant FA9550-04-1-0341 and by the Department of Defense National Security Science and Engineering Faculty Fellowship.

8:52AM MB.00005 Investigation of the one-point statistics of the scalar dissipation determined from scalar fields in large-eddy simulation , ROBERT KNAUS, CARLOS PANTANO, University of Illinois at Urbana-Champaign, JOSEPH OEFELEIN, Sandia National Laboratories — In large-eddy simulation (LES), molecular scalar mixing is completely modeled since it generally occurs at scales below the cutoff. In this case, only the resolved scalar field is simulated. In many situations the statistics of the resolved scalar fields are accurate, e.g., if the supply of unmixed fluids is provided through boundary conditions. In combustion, there are usually subgrid processes, e.g., flames, controlling the rate of chemical conversion whose coupling with the turbulence can not be resolved completely in LES. If the subgrid process is strongly sensitive to the rate of mixing, as in nonpremixed combustion, one question of interest is to what extent can one recover the true statistics of mixing from those available in the LES fields so that the accuracy of the coupling with combustion can be improved. We investigate theoretically the relationship between the resolved and true scalar dissipation and propose a framework to recover the missing one-point statistical information.

9:05AM MB.00006 Simulation of anisotropic pollution dispersion in urban environments[1] , RAFAEL IZARRA-GARCIA, Universitaet Siegen — The prediction accuracy of air flow in urban environments is strongly dependent on the selection of the turbulence model for Reynolds stresses. In case of pollution dispersion simulations, a model for the turbulent scalar fluxes is also needed and it is usually performed with the simple isotropic gradient diffusion assumption. In the present work, the influence of advanced anisotropic models for the turbulent scalar fluxes is studied and compared with two wind tunnel experiments (2D and 3D). Five anisotropic algebraic flux models, two second moment models and the simple gradient diffusion model (at various Sct) were implemented in FLUENT 6.3 and compared using statistical performance measures. The results showed general good predictions for most scalar flux models with unexpected poor improvements of the anisotropic models over the isotropic and a lack of model coefficient calibration for pollution dispersion applications. Finally, simulation improvements, sensitivity of model coefficients and several parameters and assumptions needed for this application are discussed.

[1]DAAD Stipend KN:A/05/12602.

9:18AM MB.00007 DNS and RANS Modeling of Dispersion in the Wake of an Obstacle , DAVID PHILIPS, Stanford University, RICCARDO ROSSI, Universita di Bologna, GIANLUCA IACCARINO, Stanford University — We present a numerical study of the dispersion of a passive scalar in turbulent separated flows to establish the predictive capabilities of algebraic flux models against the standard eddy-diffusivity representation. The scalar dispersion from a point source over a wavy wall is initially investigated to carefully evaluate scalar flux models through comparisons with DNS data. The roof-top release of a passive plume from a wall-mounted cube in a turbulent boundary layer is then presented to demonstrate that algebraic models can also be applied successfully to atmospheric dispersion at street-scale. Despite the questionable validity of local-equilibrium conditions, the numerical experiments show that algebraic models provide a significant improvement for scalar dispersion simulations of complex flows with respect to the standard eddy-diffusivity model.

9:31AM MB.00008 Lag-model for dissipation of scalar variance in LES[1] , SERGEI CHUMAKOV, Stanford University — Modeling the dissipation term in transport equation for the subgrid-scale (SGS) scalar variance is of high importance in LES of reacting flows. We propose a new approach to model the dissipation which is based on averaging of the source term for the SGS variance along LES Lagrangian trajectories. The averaging is performed backwards in time using a particular weight function with the peak at some non-zero characteristic time T, the "lag" time. The method is based on the notion of the non-zero cascade time scale, i.e., the production and dissipation terms are not assumed to be correlated spatially but rather temporally along LES Lagrangian trajectories. The approach results in the addition of one extra transport equation to the system, bringing the total number of auxiliary equations to two. *A priori* tests show reasonably good prediction of the SGS variance dissipation rate. Results of an *a posteriori* test for a reacting flow will be shown.

[1]Supported by NASA.

9:44AM MB.00009 A posteriori analysis of numerical errors in transport equation models for subfilter scalar variance[1] , COLLEEN M. KAUL, VENKAT RAMAN, The University of Texas at Austin — Conserved-scalar based large eddy simulations (LES) of non-premixed combustion require accurate models for subfilter scalar variance. Because most subfilter variance models depend strongly on the smallest resolved scales, they are susceptible to large numerical errors when calculated using finite difference methods. To evaluate models based on solution of a variance transport equation, errors must be assessed in a dynamic framework. The current work presents the results of a versatile a posteriori analysis method using modified wavenumbers to emulate finite difference errors in a pseudospectral code. Filtered scalar and variance fields corresponding to schemes of varying accuracy are evolved along with DNS velocity and scalar fields. This approach permits discretionary inclusion of various error sources, allowing detailed characterization of numerical and modeling issues in transport equation based subfilter variance prediction.

[1]C.M.K. gratefully acknowledges support of the NSF GRFP.

9:57AM MB.00010 Pressure Effects in Compressible Flows at the Rapid Distortion Limit , REBECCA BERTSCH, SHARATH S. GIRIMAJI, Texas A&M University — We investigate the role of pressure in rapidly sheared homogeneous compressible turbulence using the Rapid Distortion Theory (RDT). Our objective is to develop a simple dynamical model that reproduces the 3-stage turbulent kinetic energy evolution observed in compressible RDT calculations. The three stages of turbulent kinetic energy evolution are: (1) initial growth at a rate higher than equivalent incompressible flow; (2) an intermediate period of stabilization where the turbulent kinetic energy grows very slowly or not at all; and (3) final period of growth at a rate similar to incompressible flows. The dynamical model is a function of gradient and turbulent Mach numbers and attempts to shed light on the pressure-strain correlation process in compressible flows.

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MC Turbulence Simulations VI 101C

8:00AM MC.00001 The Effect of the Prandtl Number on the Turbulent Wake in a Stratified Fluid , MATTHEW DE STADLER, SUTANU SARKAR, University of California San Diego, KYLE BRUCKER[1], Science Applications International Corporation — Direct Numerical Simulations are employed to study the effect of the Prandtl number on the towed wake in a stratified fluid. Historically, the majority of stratified wake simulations are performed at a Prandtl number of 1 due to the high numerical cost of simulating larger Prandtl numbers. This makes the assumption that the Prandtl number makes only a small difference in the behavior of the wake. It is well known that the Prandtl number impacts the scale of fluctuations but it is not known how the coupling between kinetic and potential energy at different Prandtl numbers will impact the lifetime of the wake. These numerical simulations were designed to improve understanding of this aspect of the wake evolution. Data is presented for a number of mean and turbulent quantities such as the wake width, wake height, peak defect velocity, velocity fluctuations and turbulent fluxes. Simulations were conducted at a Reynolds number of 10,000 for a range of Prandtl numbers: 0.2, 1, 7, where 7 is a reasonable value for the ocean.

[1]formerly at University of California San Diego

8:13AM MC.00002 Dynamics of a Stratified Layer with Horizontal Shear , ERIC AROBONE, Main Author, APS Member, SUTANU SARKAR, Academic Adviser, APS Fellow, ALDEN KING, Research Collaborator, Computer Science PhD student, SCOTT BADEN, Research Collaborator, Computer Science Faculty — Direct Numerical Simulations of a temporally evolving uniformly stratified layer with horizontal shear provide insight into the dynamics of common oceanographic and atmospheric flows. The evolution of the stratified horizontal shear layer is investigated along with the importance of coherent vortical structures to Reynolds stresses, dissipation, correlations, spectra, and energy budgets. Novel vortex eduction techniques are employed to isolate coherent structures from the incoherent background flow. Additionally, the effect of rotation is examined with Rossby numbers appropriate for sub-mesoscale flows.

8:26AM MC.00003 Direct Numerical Simulations of unstratified and stratified wakes at Re=50,000 , KYLE BRUCKER, SUTANU SARKAR, UCSD — Direct numerical simulations (DNS) of axisymmetric wakes with and without initial net momentum are performed at Re=50,000 on a grid with approximately 2 billion grid points. The present study focuses on this difference in the presence of stratification and attempts to elucidate the effects of buoyancy. Similarities and differences are characterized by the evolution of maxima, area integrals and spatial distributions of mean and turbulence statistics. Buoyancy allows a wake to survive longer in a stratified fluid by reducing the correlation responsible for the mean-to-turbulence energy transfer in the vertical direction. This effect is especially important in the case with zero initial net-momentum because it allows regions of positive and negative momentum to become decoupled in the vertical direction and decay with different rates. The role of internal waves in the energetics is determined and it is found that they are responsible for sustaining turbulence at the wake periphery long after the shear production has subsided. The non-equilibrium region of the Re =50,000 wake is found to exhibit a time span when, although the turbulence is strongly stratified as indicated by small Froude number, the turbulent dissipation rate exhibits inertial scaling.

8:39AM MC.00004 Numerical simulations of the Lorentz force flowmeter , AXELLE VIRE, BERNARD KNAEPEN, Physique Statistique et des Plasmas, Universite Libre de Bruxelles, B-1050 Brussels, Belgium, ANDRE THESS, Fakultat fur Maschinenbau, Technische Universitat Ilmenau, P.O. Box 100565, 98684 Ilmenau, Germany — We investigate the turbulent flow of a liquid metal in a circular pipe under the influence of a localized magnetic field. The magnetic system consists in one or several coils wrapped around the pipe. The electric current in the coils generates a magnetic field that interacts with the velocity of the flow. Eddy currents are thus induced in the flow, and create a Lorentz force. In previous works, we showed that the Lorentz force acting on a coil is proportional to the mean velocity of the flow. Therefore, the measurement of this force allows an accurate determination of the mean flow rate. Here, we consider complex distributions of the magnetic field by using multiple coils, and analyse their influence on the measurement. The influence of some parameters of the coils system, such as the coil radius, is also addressed. The results are based on numerical computations performed with a second-order collocated finite volume method.

8:52AM MC.00005 Direct numerical simulation of magnetohydrodynamic flow in a toroidal duct , STIJN VANTIEGHEM, Universite Libre de Bruxelles, THOMAS DE MAET, Universite Catholique de Louvain, BERNARD KNAEPEN, VICENT MOUREAU, CORIA, CNRS UMR6614 — Magnetohydrodynamics studies the interaction between the motion of electrically conducting fluids and magnetic fields. When the magnetic Reynolds number of the flow is small, which is typical for laboratory-scale experiments and many industrial applications, we can invoke the quasi-static approximation, which states that the induced magnetic field is negligible compared to the externally applied one. In this work, we consider numerical simulations of quasi-static MHD duct flow in a toroidal duct of square cross-section. The choice of this particular geometry was inspired by a recent experimental investigation (P. Moresco and T. Alboussière, J. Fluid. Mech. 2004) of the instability of the Hartmann layers. The scope of this work is however a more moderate regime in terms of Reynolds and Hartmann number, so that the Coriolis force will also play an important role, as expressed by the Elsasser number. We investigate the transition of the flow and the turbulent statistics as a function of the aforementioned parameters.

9:05AM MC.00006 Spectral study of anisotropic magnetohydrodynamic turbulence , BOGDAN TEACA, Universite Libre de Bruxelles, MAHENDRA VERMA, Indian Institute of Technology Kanpur, DANIELE CARATI, Universite Libre de Bruxelles — A spectral analysis of anisotropic magneto-hydrodynamic turbulence, in presence of a constant magnetic field, is presented using direct numerical simulations. A method of decomposing the spectral space into ring structures is presented and the energy transfers between such rings are studied. This decomposition method takes into account the angular dependency of transfer functions in anisotropic systems, while it allows to recover easily the known shell-to-shell transfers in the limit of isotropic turbulence. For large values of the constant magnetic field, the dominant energy transfers appear to be in the direction perpendicular to the mean magnetic field. The linear transfer due to the constant magnetic also appear to be important in redistributing the energy between the velocity and the magnetic fields.

9:18AM MC.00007 Direct numerical and large eddy simulations of decaying magnetohydrodynamic turbulence at low magnetic Reynolds number , ABHILASH CHANDY, University of Akron, STEVEN FRANKEL, Purdue University — A series of direct numerical simulations (DNS) is performed for decaying homogeneous magnetohydrodynamic (MHD) turbulence at low magnetic Reynolds number ($Re_m << 1$) with different strengths of magnetic field. The initially-isotropic turbulence problem has a Taylor scale Reynolds number (Re_λ) of 220. Comparisons of decay rates, energy spectra and even higher-order statistics such as structure functions and skewness factors are made between the varying magnetic field cases. Furthermore, the phenomenon of anisotropy, that is developed due to the introduction of the magnetic field is investigated by comparing anisotropy coefficients based on velocities and their gradients. Large eddy simulations (LES) using the classical non-dynamic Smagorinsky model are also conducted for the highest magnetic field case and results are in excellent agreement with the corresponding DNS.

9:31AM MC.00008 Direct Numerical Simulation for the MHD Homogeneous Shear Turbulence with the Several Parameter Sets , MASAYOSHI OKAMOTO, DAISUKE HAMADA, Department of the Mechanical Engneeing, Shizuoka University — The MHD homogeneous shear turbulent flows with the several initial ratios of the magnetic energy to the kinetic energy, magnetic Prandtl number, shear rates and initial profiles of the cross helicity are investigated by means of the direct numerical simulation. The transformation term between the kinetic and magnetic energy, which is closely connected with the Lorenz force, plays an important role of the interaction between the velocity and magnetic fields. In the cases of the large initial energy ratio and high magnetic Prandtl number, the magnetic energy is converted into the kinetic one and from the result of the energy spectrum budget this converting phenomenon occurs over the wide length scale. On the other hand, the inverse conversion is caused in the large scale in the case of the small initial energy ratio, small magnetic Prandtl number and the high shear rate. The difference among the initial cross helicity has an influence on the time development of the mean quantities immediately after the calculation start.

9:44AM MC.00009 Rayleigh-Taylor turbulence in presence of stratification[1] , LUCA BIFERALE, MAURO SBRAGAGLIA, ANDREA SCAGLIARINI, Dept. Physics, University Tor Vergata Roma, FEDERICO TOSCHI, Technische Universiteit Eindhoven — We present numerical results on 2 dimensional Rayleigh-Taylor turbulence in presence of density stratification, up to Atwood =0.4. Numerical algorithm is based on a fully consistent Thermal Lattice Boltzmann code for an ideal gas. We study both how the mixing layer depends on stratification and small-scale temperature, density and velocity fluctuations, confirming a Bolgiano scaling scenario for two dimensional RT turbulence.

[1]ICTR Collaboration

9:57AM MC.00010 Prediction of pressure fluctuations in turbulent flows using the immersed boundary method[1] , SEONGWON KANG, GIANLUCA IACCARINO, FRANK HAM, PARVIZ MOIN, Center for Turbulence Research, Stanford University — The immersed boundary (IB) method has been widely used to model flow problems in complex geometries. We investigate the capability of the IB method to predict wall pressure fluctuations in turbulent flows. We introduce a new numerical treatment of the cells crossed by the IB that ensures mass consrvation and provides accurate evaluation of the wall pressure. The present approach has been successfully validated through computations of the space-time correlations of the wall-pressure fluctuations. Compared to the original IB method (Fadlun et al., 2000), the present approach shows better agreement with the standard DNS results. When applied to turbulent flow around an airfoil, the computed flow statistics - the mean/RMS and power spectra of the wall pressure - are in good agreement with the LES performed on body- fitted mesh and experiment (Roger and Moreau, 2004).

[1]Supported by Stanford DOE ASC program.

Tuesday, November 24, 2009 8:00AM - 9:57AM –
Session MD Chaos and Fractals II 101D

8:00AM MD.00001 Sensitivity of LCS identification to flow field resolution and random errors , ALI B. OLCAY, UWP, TAIT S. POTTEBAUM, USC, PAUL S. KRUEGER, SMU — Ridges in the finite-time Lyapunov exponent (FTLE) fields obtained from fluid flows provide separatrices that are transport barriers. These ridges, called Lagrangian coherent structures (LCSs), can be used to identify vortex boundaries and study mixing processes in complex, time-dependent flows. The accuracy of LCS identification is inherently dependent on the accuracy of the underlying velocity field used to compute the FTLE field. To quantify the effect of velocity field errors, this study considers the two canonical flows: a steady 2D vortex pair and an axisymmetric vortex ring generated by a starting jet. The velocity field for the vortex pair was determined analytically while the vortex ring flow was obtained from CFD for jet $Re = 1000$. Velocity field errors were introduced by smoothing and sub-sampling the data (mimicking DPIV interrogation window size effects) and adding random noise. The results show that the LCS location can be shifted as much as $0.2D$ when sub-sampling the velocity field from a resolution of $0.01D$ to $0.125D$ (D is the vortex diameter) and the effect of noise on the mean LCS location is small compared to the resolution effect.

8:13AM MD.00002 Characterization of Mixing Using Experimentally Derived Velocity Fields and Derived Lagrangian Coherent Structures , DOUGLAS BOHL, NARATIP SANTITISSADEEKORN, ERIK BOLLT, Clarkson University — In this work a flat rectangular plate is rotated along its long axis and parallel to the z-axis of a circular cylinder. The blade position is varied with respect to the cylinder wall to allow investigation of the effect of the no slip boundary on the flow structure and mixing field. The cylinder is filled with viscous Newtonian fluids and driven at low Reynolds numbers (8-100). Particle Image Velocimetry is used to measure the velocity in the plane perpendicular to the rotation of the plate (i.e. in the r-θ plane of the cylinder). The experimental velocity field is used to 1. Determine numerically the motion of 100,000 simulated zero mass particle tracers for up to 25 cycles of the blade and 2. Calculate the Lagrangian Coherent Structures (LCS) of the flow field. Mixing rates and length scales are found using the particle tracers. Results show that the fluid is segregated into distinct regions with limited interaction by the ridges in the short term LCS field. The results also show that if the Lagrangian Coherent Structures are calculated for long times the resulting LCS field is similar to that of the particle distribution field for the same time number of rotations.

8:26AM MD.00003 Identifying Exact Coherent Structures in 2D Turbulence: Experiments and Simulations , MICHAEL SCHATZ, JON PAPROCKI, CHRISTOPHER LESESNE, JAMES ANDREWS, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology — Recent theoretical advances suggest ways to find unstable exact Navier Stokes solutions that capture many features of coherent structures, which have long been observed in turbulent flow. At present, it remains unknown whether these solutions, termed Exact Coherent States, can describe observations of turbulent flow in laboratory experiments. We describe our experimental and numerical investigations, which search for unstable solutions in quasi-2D flows driven by electromagnetic forces. In the experiments, time series of velocity fields are obtained from images of the visualized flow. In the simulations, long time series of velocity fields are calculated for flows with forcing similar to that in the experiments. The velocity field data from both experiments and simulations are used to construct recurrence plots that provide evidence for the existence of unstable periodic orbits.

8:39AM MD.00004 Characterizing the Chaotic Degrees of Freedom of High-Dimensional Fluid Convection[1] , ALIREZA KARIMI, MARK PAUL, Virginia Tech — The variation of the Lyapunov exponent spectra and fractal dimension with system parameters can yield fundamental insights into the nature of spatiotemporal chaos. We explore this numerically for two systems: the Lorenz 96 model and Rayleigh-Benard convection. The Lorenz 96 model is a phenomenological model that captures important features of atmosphere dynamics. We compute the fractal dimension as a function of system size and external forcing for very long times and over many initial conditions. When varying system size we find extensive chaos with significant deviations from extensivity for small changes in system size and also the power-law growth of the dimension with increasing forcing. We use large-scale parallel numerical simulations to study chaotic Rayleigh-Benard convection for experimentally accessible conditions. We compute the variation of the fractal dimension with system size, Prandtl number, and Rayleigh number. Using statistical properties of the Lyapunov exponents and Lyapunov vectors we connect these features with the dynamics of the flow field pattern.

[1]NSF Award CBET-0747727, NSF Teragrid, Virginia Tech's Advanced Research Computing Center.

8:52AM MD.00005 Dynamically Evolving Topology in Spatiotemporal Chaos , NICHOLAS T. OUELLETTE, DOUGLAS H. KELLEY, Yale University — Recent advances in Lagrangian measurements have allowed the robust experimental location of the hyperbolic and elliptic stagnation points in two-dimensional incompressible flow. Here, we extend these techniques to study the dynamics of the stable and unstable manifolds of the hyperbolic points as our experimental quasi-2D electromagnetically driven flow evolves in time. We compare results from nearly stationary flows, where the hyperbolic points move only slightly, and spatiotemporally chaotic flows, where hyperbolic and elliptic points can be created or annihilated in pairs. This work is supported by the National Science Foundation.

9:05AM MD.00006 Uncertainties in Lagrangian mixing , WENBO TANG, ALEX MAHALOV, Arizona State University — Lagrangian Coherent Structures have been discovered to be the building blocks of chaotic mixing in turbulent flows and mathematical tools have been developed to extract the invariant manifolds that highlight Lagrangian mixing. These mathematical tools are based on deterministic velocity fields and it is unclear how random processes can modify chaotic mixing. In this talk we discuss archetypical geophysical flow examples embedded in an environment of Gaussian white noise. We examine how the deterministic nonlinear background flow fields alter the Gaussian statistics and the consequences of stochastic processes on Lagrangian mixing.

9:18AM MD.00007 A Chaotic Periodically Reoriented Irrotational Flow: Experiments, Theory, and Applications to Geophysical Transport , GUY METCALFE[1], DANIEL LESTER[1], MURRAY RUDMAN[2], KLAUS REGENAUER-LIEB[3], MIKE TREFRY[4], ALISON ORD[3], BRUCE HOBBS[3], CSIRO Australia, PANDURANG KULKARNI, University of California, Santa Barbara, PARK KWAN FUNG, University of New South Wales, ZHURUI XU, Universite Pierre et Marie Curie, JEFF MORRIS, City College of New York — A source-sink pair in a Hele-Shaw cell generates an irrotational dipole flow. In a disk with 360 wells around its periphery and a rotatable manifold controlling which well pairs are open, we have created a periodically reoriented dipole flow which is an open chaotic dynamical system with properties controlled by the reorientation angle and duration of flow. Despite being open the flow can have island regions where fluid that starts in the disk remains there forever. Theory and experiments determine the island existence boundary in control parameter space, the variation in island size, and bifurcations. We also briefly describe possible applications to problems in geophysical transport.

[1]Materials Science and Engineering
[2]Mathematics and Information Science
[3]Exploration and Mining; University of Western Australia
[4]Land and Water; University of Western Australia

9:31AM MD.00008 The Dynamics of Ratcheting States in Cellular Flames , MICHAEL GORMAN, University of Houston — Cellular flames form ordered states of two concentric rings of brighter, hotter cells, separated by darker, cooler cusps and folds. In ratcheting states one or both rings of cells of cells rotate slowly ($\sim$1 deg/sec), speeding up and slowing down in a manner characteristic, which depends on the numbers of cells in the inner and outer rings and the degree of coupling between the two rings. We present measurements of the velocities of 4 such states and video clips of the motions of 20 other ratcheting states. The characteristics of these states have not yet been explained. The nature of ratcheting motion has not yet been described in the context of bifurcations with symmetry.

9:44AM MD.00009 Cryptography with Chaos and Shadowing[1] , NEJIB SMAOUI, ALI KANSO, Kuwait University — A novel approach to encrypt a message using chaos and shadowing is presented. The approach is based on two steps: First, a numerical chaotic orbit of the logistic map is used in the shadowing algorithm of Smaoui & Kostelich [Intern. J. Computer. Math. (1998) 70] to show that there exists a finite number of true orbits that shadow the numerical orbit. Then, Baptista's algorithm [Phys. Lett. A (1998) 240] is used on the finite number of maps to encrypt each character of the message. It is shown that the use of the shadowing method in the encryption process enhances the security level.

[1]This work was supported by Kuwait University Research Grant No. SM04/07.

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session ME Biofluids IX: Nasal, Lungs and Other Organs 101E

8:00AM ME.00001 Flow Visualization Experiments in a 4:1 Scale Model of the Canine Nasal Cavity , MICHAEL HARGATHER, MICHAEL LAWSON, GARY SETTLES, Penn State Gas Dynamics Lab — An anatomically-correct 4:1 scale model of the canine nasal cavity is used to study flow patterns in the complex nasal airways through dye-streak flow visualization. The nasal cavity geometry was obtained from magnetic resonance imaging (MRI) scans and the model was constructed in sections from a transparent material using a rapid prototyping technique. We believe this model represents the first anatomically-realistic reproduction of the canine nasal cavity, allowing the nasal flowfield to be experimentally studied at a level of detail not previously possible. Olfactory and respiratory flows are observed to take separate paths through the nasal cavity. Respiratory flow through the maxilloturbinates completely bypasses the olfactory region, which amounts to a "side-sampler." A single airway conducts airflow into the olfactory region, whence it slowly filters forward and eventually exits the nasal cavity. The residence time of airflow in the olfactory region varies significantly depending on the specific flowpath taken. The results compare well with computational fluid dynamics (CFD) simulations performed using the same nasal geometry.

8:13AM ME.00002 Odorant Transport and Deposition in the Canine Nasal Airways , MICHAEL LAWSON, BRENT CRAVEN, GARY SETTLES, ERIC PATERSON, Pennsylvania State University — The canine nose functions similar to a chromatograph that imposes odorant-specific deposition patterns upon the thin mucus layer covering the nasal cavity. Here we use an anatomically-correct computational fluid dynamics (CFD) model to study airflow and odorant transport from the external environment through the nasal airways to the olfactory receptor layer beneath the mucus. The results show that deposition patterns are primarily influenced by the intricate olfactory flowfield and the odorant solubility in the mucus layer. Highly-soluble odorants are quickly absorbed near the entrance to the olfactory region, and thus do not reach the periphery with significant concentrations. In contrast, insoluble odorants are deposited more evenly and may even exit the olfactory region without being completely absorbed. Predicted odorant deposition patterns correspond with the anatomical organization of olfactory receptors known to occur in keen-scented (macrosmatic) mammals, providing a mechanism that helps explain the excellent olfactory acuity of the dog.

8:26AM ME.00003 Alveolar mechanics using realistic acinar models[1] , 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, Physiology, and Biomedical Engineering, The University of Iowa — Accurate modeling of the mechanics in terminal airspaces of the lung is desirable for study of particle transport and pathology. The flow in the acinar region is traditionally studied by employing prescribed boundary conditions to represent rhythmic breathing and volumetric expansion. Conventional models utilize simplified spherical or polygonal units to represent the alveolar duct and sac. Accurate prediction of flow and transport characteristics may require geometries reconstructed from CT-based images and serve to understand the importance of physiologically realistic representation of the acinus. In this effort, we present a stabilized finite element framework, supplemented with appropriate boundary conditions at the alveolar mouth and septal borders for simulation of the alveolar mechanics and the resulting airflow. Results of material advection based on Lagrangian tracking are presented to complete the study of transport and compare the results with simplified acinar models. The current formulation provides improved understanding and realization of a dynamic framework for parenchymal mechanics with incorporation of alveolar pressure and traction stresses.

[1]This work is sponsored by NIH Grants R01-EB-005823 and R01-HL-064368.

8:39AM ME.00004 On boundary-layer and free-shear resistances in the human airways[1] , JIWOONG CHOI, CHING-LONG LIN, The University of Iowa, MERRYN TAWHAI, The University of Auckland, ERIC HOFFMAN, The University of Iowa — The airway resistance has been reported to be greater on expiration than inspiration. To understand the underlying mechanism, we perform large eddy simulation of airflow in the 3D CT-resolved 7-generation airways constrained by physiologically-consistent lobar ventilation. The dimensionless viscous pressure drops in all the airway segments exhibit a similarity behavior proportional to $(ReD/L)^n$ with the average optimal values of 1.4 and 1.6 for inspiration and expiration, respectively, where Re is the Reynolds number, and D and L are the respective average diameter and length of an airway segment. It is found that the dissipations in the boundary layer as well as the free-shear core flow contribute to the airway resistance, thus the n value. Flow is partitioned to examine the roles played by the boundary layer and the free-shear flow, respectively. A hypothesis is proposed to explain higher airway resistance on expiration.

[1]The work is sponsored by NIH Grants R01-EB-005823 and R01-HL-064368.

8:52AM ME.00005 Steady propagation of Bingham plugs in 2D channels , PARSA ZAMANKHAN, SHUICHI TAKAYAMA, JAMES GROTBERG, University of Michigan - Department of Biomedical-Engineering — The displacement of the yield-stress liquid plugs in channels and tubes occur in many biological systems and industrial processes. Among them is the propagation of mucus plugs in the respiratory tracts as may occur in asthma, cystic fibrosis, or emphysema. In this work the steady propagation of mucus plugs in a 2D channel is studied numerically, assuming that the mucus is a pure Bingham fluid. The governing equations are solved by a mixed-discontinuous finite element formulation and the free surface is resolved with the method of spines. The constitutive equation for a pure Bingham fluid is modeled by a regularization method. Fluid inertia is neglected, so the controlling parameters in a steady displacement are; the capillary number, Ca, Bingham number ,Bn, and the plug length. According to the numerical results, the yield stress behavior of the plug modifies the plug shape, the pattern of the streamlines and the distribution of stresses in the plug domain and along the walls in a significant way. The distribution along the walls is a major factor in studying cell injuries. This work is supported through the grant NIH HL84370.

9:05AM ME.00006 Flow through flexible cylinders inspired by the endothelial glycocalyx , LAUREN COOPER, DANIEL FOVARGUE, LAURA MILLER, The University of North Carolina at Chapel Hill — Inspired by the recent shift in hypertension research, we present a new computational model to better examine blood flow induced shear stress in the endothelial surface layer (ESL). The ESL is the luminal side barrier between blood and the endothelial cells that line the vessel wall and has been of interest due to its function as a mechanotransducer.[1] Further, it is believed that shear stress seen by the ESL, induced by blood flow, is converted to chemical responses such as blood pressure regulation. We utilize the Immersed Boundary method to simulate blood flow through a vessel and examine the shear stress at the ESL over different heights and flexibilities. We compare our results in the Reynolds number regime of a canine capillary with previous computational models[2] and experimental results.

[1]Squire, J. M., Chew, M., Nneji, G., Neal, C., Barry, J. & Michel, C. C., 2001. Quasi-periodic substructure in the microvessel endothelial glycocalyx: a possible explanation for molecular filtering? J. Struct. Bio. 136, 239-255.
[2]Weinbaum, S., Tarbell, J., Damiano, E., 2000. The Structure and Function of the Endothelial Glycocalyx Layer. Pflügers Arch. – Eur. J. Physiol. 440, 653–666.

9:18AM ME.00007 High-frequency self-excited oscillations in collapsible tube flows[1] , ROBERT J. WHITTAKER, SARAH L. WATERS, University of Oxford, UK, OLIVER E. JENSEN, University of Nottingham, UK, JONATHAN BOYLE, MATTHIAS HEIL, University of Manchester, UK — Experiments show that steady flow along an elastic-walled tube can become unstable to large-amplitude oscillations involving both the tube wall and the fluid. We consider a "Starling resistor" setup - a finite length elastic tube attached to rigid end sections, through which an axial flow is driven by either a steady flux at the downstream end or a steady pressure drop between the ends. We present a theoretical analysis of small-amplitude high-frequency long-wavelength oscillations. We first consider the fluid mechanics (with prescribed wall oscillations) and then the solid mechanics (to derive an appropriate tube law) in isolation. The two strands are then combined to investigate the full 3D fluid–structure interaction problem for self-excited oscillations. We determine the form of the normal modes and obtain expressions for the growth rate and frequency of the oscillations. The predictions from our modeling are compared with numerical simulations performed using the oomph-lib library.

[1]Supported by EPSRC

9:31AM ME.00008 Control volume based hydrocephalus research; a phantom study , BENJAMIN COHEN, RPI, ABRAM VOORHEES, Siemens Medical Imaging, JOSEPH MADSEN, Harvard Medical School, TIMOTHY WEI, RPI — Hydrocephalus is a complex spectrum of neurophysiological disorders involving perturbation of the intracranial contents; primarily increased intraventricular cerebrospinal fluid (CSF) volume and intracranial pressure are observed. CSF dynamics are highly coupled to the cerebral blood flows and pressures as well as the mechanical properties of the brain. Hydrocephalus, as such, is a very complex biological problem. We propose integral control volume analysis as a method of tracking these important interactions using mass and momentum conservation principles. As a first step in applying this methodology in humans, an *in vitro* phantom is used as a simplified model of the intracranial space. The phantom's design consists of a rigid container filled with a compressible gel. Within the gel a hollow spherical cavity represents the ventricular system and a cylindrical passage represents the spinal canal. A computer controlled piston pump supplies sinusoidal volume fluctuations into and out of the flow phantom. MRI is used to measure fluid velocity and volume change as functions of time. Independent pressure measurements and momentum flow rate measurements are used to calibrate the MRI data. These data are used as a framework for future work with live patients and normal individuals. Flow and pressure measurements on the flow phantom will be presented through the control volume framework.

9:44AM ME.00009 The Relation between Peristaltic and Segmental Contraction, Mixing, and Absorption in the Small Intestine , GINO BANCO, JAMES BRASSEUR, YANXING WANG, AMIT AILIANI, THOMAS NEUBERGER, ANDREW WEBB, Penn State Univ — The physiology and mechanics of the small intestine originates with lumen-scale fluid motions generated by enterically controlled muscle wall contractions. Although complex in appearance, we have shown with principle component decomposition of gut motion from a rat model that simpler component structure may integrate to produce basic peristaltic and segmental motions. To couple these measured modes with fluid mixing and nutrient absorption we have developed 2-D and axisymmetric models of the gut using the lattice-Boltzmann framework with scalar and second order moving boundary conditions. Previous models indicated that peristalsis is detrimental to absorption and therefore that gut motility is likely bimodal, transitioning between peristalsis and segmental modes to optimize the transport of chyme vs. nutrient absorption. However we have since discovered that more complex control is possible due to potential transitions between "trapped" vs. "nontrapped" peristaltic fluid motions, depending on occlusion ratio. These transitions lead to an important distinction between 2-D and axisymmetric models and indicate that gut motility may be more finely controlled than previously thought. [Supported by NSF]

9:57AM ME.00010 An Integrative Model of Excitation Driven Fluid Flow in a 2D Uterine Channel , CHARLES MAGGIO, LISA FAUCI, JOHN CHRISPELL, Tulane University — We present a model of intra-uterine fluid flow in a sagittal cross-section of the uterus by inducing peristalsis in a 2D channel. This is an integrative multiscale computational model that takes as input fluid viscosity, passive tissue properties of the uterine channel and a prescribed wave of membrane depolarization. This voltage pulse is coupled to a model of calcium dynamics inside a uterine smooth muscle cell, which in turn drives a kinetic model of myosin phosphorylation governing contractile muscle forces. Using the immersed boundary method, these muscle forces are communicated to a fluid domain to simulate the contractions which occur in a human uterus. An analysis of the effects of model parameters on the flow properties and emergent geometry of the peristaltic channel will be presented.

Tuesday, November 24, 2009 8:00AM - 9:57AM –
Session MF Microfluidics: Electrokinetics and Transport 101F

8:00AM MF.00001 Numerical and experimental study of dispersion dynamics in isotachophoresis , GIANCARLO GARCIA, MORAN BERCOVICI, JUAN G. SANTIAGO, Stanford University — Isotachophoresis (ITP) is a separation and preconcentration technique used in a variety of life science and analytical chemistry applications. Under ideal ITP conditions, sample ions focus in a narrow interface region (1-10 μm) between leading and terminating electrolytes. In practice, however, the associated electric field gradients at this interface give rise to non-uniform electroosmotic flow (EOF) and associated strong internal pressure gradients. Conductivity gradients also couple with electric fields to yield electrohydrodynamic body forces. Together, these forces disperse the ITP interface and reduce the sensitivity and resolution of ITP-based assays. Despite its importance in ITP, there has been surprisingly little research into the underlying physical mechanisms of dispersion. We performed a numerical and experimental study of dispersion dynamics in ITP using two-dimensional (axi-symmetric), time-dependent simulations of fluid flow, diffusion, and electromigration. We validated our models with controlled experiments in circular capillaries and used simulations to develop general scaling relationships. We observe localized focusing of the analyte in either near-axis or near-wall regions; and the degree to which these conditions are favored is a strong function of the axial location of the ITP interface. Our goal is to develop an area-averaged model for rapid prediction of the effects of EOF in experiments.

8:13AM MF.00002 Concentration boundary layers in osmotic membrane transport processes[1] , KAARE JENSEN, TOMAS BOHR, HENRIK BRUUS, Center for Fluid Dynamics, 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 [1,2]. Previous experimental as well as theoretical works have focused on the case where the solution on both sides of the membrane remain well-mixed due to an external stirring mechanism. We present a theoretical investigation the effects of concentration boundary layers on the efficiency of osmotic pumping processes in the absence of external stirring i.e. when the stirring is provided by the osmotically generated flow itself. For such systems, we show that no well defined boundary layer thickness exist and that the reduction in concentration can be estimated by a surprisingly simple mathematical relation valid across a wide range of geometries and Péclet numbers.

[1] T.J.Pedley, Q. Rev. Biophys., 1983, **16**, 115
[2] K.H.Jensen et al., Lab Chip, 2009, **9**, 2093

[1]This work was supported by the Danish National Research Foundation, Grant No. 74.

8:26AM MF.00003 Induced Charge Electrokinetics Over "Controllably Contaminated" Surfaces: The Effects of Dielectric Thin Films and Surface Chemistry on Slip Velocity , ANDREW PASCALL, TODD SQUIRES, University of California, Santa Barbara — Microfluidics has renewed interest in utilizing electrokinetics (EK) for transporting fluids on small scales, and has subjected EK theories and understanding to new challenges. For example, induced-charge electro-osmosis (ICEO), a non-linear EK effect in which an externally applied AC electric field both induces and drives a layer of charged fluid near an electrically conductive surface, could provide an on-chip means to drive high pressures with low voltage [1]. Experimental data on ICEO and related phenomena have shown that the standard theory consistently overpredicts slip velocities by up to a factor of 1000[2]. Here we present experiments in which we controllably "contaminate" the metallic surface with a thin dielectric film or Au-thiol self assembled monolayer, and derive a theory for ICEO that incorporates both dielectric effects and surface chemistry, which both act to decrease the slip velocity relative to a 'clean' metal. Data for over a thousand combinations of electric field strength and frequency, electrolyte composition, dielectric thickness and surface chemistry show essentially unprecedented quantitative agreement with our theory. [1] Squires & Bazant. J. Fluid Mech. 2004 [2] Bazant, et al. arXiv. 0903.4790

8:39AM MF.00004 Concentration polarization effects in nanochannel induced-charge electro-osmosis , MATHIAS ANDERSEN, HENRIK BRUUS, Department of Micro- and Nanotechnology, Technical University of Denmark, DTU Nanotech Building 345 East, DK-2800 Kongens Lyngby, Denmark — Concentration polarization (CP) has been observed in a variety of configurations: nanochannels, nanopores, ion-permselective membranes, and embedded conductors. It is believed, but not conclusively proven, that electric double-layer (EDL) overlap plays a significant role in CP. Consequently, further studies of fundamental electrokinetic effects related to CP are needed. We present theoretical and numerical studies of CP effects near an un-biased conductor placed in a straight nanoslit. In our model we combine the full Poisson-Nernst-Planck equations with Helmholtz-Smoluchowski slip, whereby we can construct large computational domains and at the same time resolve the EDL in the vicinity of the conductor. Based on our results we try to identify the basic physical mechanisms that lead to the CP. We show that when subjected to an external bias the axial symmetry is broken by the electro-osmotic flow. This leads to nonlinear interactions between flow, electric potential, and charged chemical species in the induced EDL on the conductor.

8:52AM MF.00005 Transient Current and Fluid Transport in Electrolyte Displacement by Electro-osmotic Flow , HSIEN-HUNG WEI, SZU-WEI TANG, CHIEN-HSIANG CHANG, National Cheng Kung University — In this work, the displacement between two different conductivity solutions by an electro-osmotic flow in a uniformly charged channel is revisited theoretically in the large Peclet number limit. A conductivity mismatch can induce an additional pressure flow due to unequal electro-osmotic slip velocities in the solutions. And yet, we argue that the notion of uniform displacement can still be valid *locally* in the vicinity of the moving concentration front. We derive a coupled set of equations for the electric current and the displacement distance and obtain an analytical solution for these equations. We find that the displacement can exhibit distinct features, depending on the range of the conductivity ratio. This is demonstrated by examining three limiting scenarios: (i) when the solution conductivities are nearly matched, (ii) the displacement by a very high conductivity solution, and (iii) the use of a very low conductivity solution in advancing the displacement. Our findings provide some insights into the zeta potential measurement using the current monitoring method and sample stacking by electro-osmotic flow with conductivity gradient. Effects of dispersion are also discussed.

9:05AM MF.00006 Charged solute transport and separation in nanochannels with surface roughness , GUOQING HU, Laboratory of Nonlinear Mechanics, Institute of Mechanics, CAS, Beijing 100190 — Newfound attention has been given to solute transport in nanochannels. Because the electric double layer thickness is comparable to characteristic channel dimensions, nanochannels have been used to separate ionic species with a constant charge-to-size ratio (i.e., electrophoretic mobility) that otherwise cannot be separated in electroosmotic or pressure-driven flow along microchannels. Surface roughness is usually inevitable during the fabrication of nanochannels. We develop a numerical model to investigate the transport and separation of charged solutes in nanochannels with hundreds of roughness elements. The solute transport patterns in rough channels are compared with those in smooth channels. The effects of surface roughness on the migration speed and retention (defined as the ratio of the solute speed to the fluid speed, used to characterize the efficiency of solute separation) of various solutes at different electrolyte concentrations are examined. Results indicate that solutes move slower in rough nanochannels than in smooth ones for both pressure-driven and electroosmotic flows. Solute separation can be significantly improved by surface roughness under certain circumstances.

9:18AM MF.00007 Ion Transport in Electric Double Layer Capacitors for Water Desalination[1] , BATYA FELLMAN, EVELYN WANG, Massachusetts Institute of Technology — Capacitive deionization is a promising method for efficient water desalination. In this approach, salt water is passed through two polarized electrodes, whereby the salt absorbs onto the electrode surface for removal. We investigated high surface area carbon-based electrode materials, including activated carbon and activated carbon cloth, for capacitive deionization. In a 1 M NaCl electrolyte, the activated carbon cloths with a surface area of 1000-2000 m^2/g exhibited specific capacitance values of approximately 40 F/g, which is an order of magnitude lower than that of state-of-the-art aqueous capacitors. We speculate that the discrepancy is related to transport limitations at the electrode-electrolyte interface. Based on these studies, we fabricated new controlled electrode geometries and surface chemistries to enable detailed studies of ion transport in the electric double layer and to understand the effect on charging times and specific capacitance. Experimental techniques, including cyclic voltammetry, chronocoulometry, and impedance spectroscopy, were used. These studies help elucidate transport mechanisms and provide insight into optimal design for effective capacitive deionization.

[1]Funded by Center for Clean Water and Clean Energy at MIT and KFUPM.

9:31AM MF.00008 Carbon dioxide and water multiphase flow in microchannels , RUOPENG SUN, THOMAS CUBAUD, Stony Brook University — We experimentally study the microfluidic formation and evolution of CO2 bubbles in distilled water at room temperature. Using silicon/glass microchannels, water and carbon dioxide are mixed in a cross-shaped focusing section. The decreasing length of the produced bubbles dissolving in water is measured along a square microchannel as a function of flow rates and inlet pressures. We calculate the rate of bubble dissolution from high-speed imaging and study the flow transition from elongated to spherical bubbles. We also investigate the micro-flow behavior of carbonated water and the interrelation between bubble nucleation and frictional pressure drop.

9:44AM MF.00009 Diffusion-limited evaporation in microchannels[1] , ADAM HOFFMAN, KENNETH BREUER, Brown University — We present measurements regarding the evaporation of a liquid from long microchannels. The channels are approximately 9000 microns long, 100 and 500 microns wide and range in depth from 1 to 22 microns. Both ends are open to the atmosphere. The evaporation of the liquid slug within the channel is measured by optically tracking the recession of the menisci from both ends. Different fluids are tested as well as different surface treatments, and the results are modeled in the context of a simple theory based on the diffusion of the vapor along the long channel. At these scales, the shape of the meniscus and its wetting behavior is shown to have a strong effect on the overall evaporation rates, and we hypothesize that this is due to the role of the transition and absorbed films that precede the visible meniscus.

[1]Supported by NSF.

Tuesday, November 24, 2009 8:00AM - 9:44AM –
Session MG Nano-Fluids II 101G

8:00AM MG.00001 A nonlinear equation for ionic diffusion in a strong binary electrolyte[1] , SANDIP GHOSAL, ZHEN CHEN, Northwestern University — The problem of the one dimensional electro-diffusion of ions in a strong binary electrolyte is considered. The mathematical description consists of a diffusion equation for each species augmented by transport due to a self consistent electrostatic field determined by the Poisson equation. If concentrations do not vary appreciably over distances of the order of the Debye length, the Poisson equation can be replaced by the condition of local charge neutrality first introduced by Planck. It can then be shown that both species diffuse at the same "ambipolar" rate with a common diffusivity.Here we derive a more general theory by exploiting the ratio of Debye length to a characteristic length scale as a small asymptotic parameter. It is shown that the concentration of either species may be described by a nonlinear integro-differential equation which replaces the classical linear equation for ambipolar diffusion but reduces to it in the appropriate limit. Through numerical integration of the full set of equations it is shown that this nonlinear equation provides a better approximation to the exact solution than the linear equation it replaces.

[1]Support from the NIH under grant R01EB007596 is gratefully acknowledged.

8:13AM MG.00002 Porescale transport phenomena in charge-selective hemofilters[1] , SUBHRA DATTA, ALBERT CONLISK, The Ohio State University — Theoretical models for hindered transport of biomolecules and electrostatic and electrokinetic phenomena in the pressure driven flow of blood simulants in structured nanoporous membranes are developed, motivated by the design requirements for a hemofilter for an implantable artificial kidney. In particular, the selectivity of charged membrane to charged biomolecules of biological interest, the inference of the pore wall surface charge density from streaming potential measurements, when electrical double layers overlap and the pore wall surface charge density is heterogeneous (e.g. due to nonuniformities in the applied surface coatings) and the coupling of intrapore phenomena with mass transfer and fluid flow upstream and downstream of the membrane are discussed. The developed theory is applied to the problem of choosing a hemofilter pore size that provides adequate retention/clearance of desirable/undesirable solutes from blood.

[1]Support from the National Institute of Biomedical Imaging and Bioengineering through NIH Grant Number R01EB008049 is acknowledged.

8:26AM MG.00003 Micropancake Growth , JAMES SEDDON, OLESYA BLIZNYUK, STEFAN KOOIJ, HAROLD ZANDVLIET, BENE POELSEMA, DETLEF LOHSE, University of Twente — Micropancakes are thin fluidic layers that form at the interface between a hydrophobic substrate and bulk liquid. To date, details of how they differ from the bulk liquid has been the subject of speculation. The current consensus is that they are gaseous, with typical heights of 1-2nm and lateral extents of microns. Here we present results of an experimental investigation into the dynamics of micropancakes. Atomic force microscopy is used to firstly confirm the existence of micropancakes before monitoring long-time effects. We find that the micropancakes grow laterally with time, thus tending to reduce the area of contact between the bulk liquid and substrate. The growth is initially directional, mediated by substrate pinning, however a surprising transition from this growth behaviour to pancake rearrangement then occurs.

8:39AM MG.00004 Applications of Needle-Like Magnets in Opto-Fluidics[1] , ALEXANDER TOKAREV, MEENA MIRDAMADI, MATTHIEU BARDET, SERHIY MALYNYCH, GEORGE CHUMANOV, JOHN BALLATO, KONSTANTIN KORNEV — Imaging in micro and nanofluidics is a challenge: the sizes of micro and nanochannels are so small that the installment of additional optical and mechanical switches is almost impossible. Another constrain is the size of the device and associated increase in viscous dissipation. We suggest manipulating the light by using the existing fiber optics and adding a functional lens which would expand/contract the light on demand. We showed that the shape of the laser beam passing through the colloid with suspended magnetic nanoparticles can be altered by varying the applied magnetic field. When the propagation of light is perpendicular to the magnetic field, this lens filled with a magnetic fluid works as a cylindrical lens focusing the light onto a line instead of onto a point. In the paper we report the experimental results on kinetics of chain formation as observed through dynamic light absorption and multiple scattering. Magnetic nanoneedles are also attractive candidates for making deformable optical lenses. We show that optically transparent solutions of nickel nanowires in ethylene glycol are responsive to weak magnetic fields. When magnetic field is applied from the top, the droplet of this solution changes its shape making a central spike at some critical field. A theoretical analysis and experimental data on this unusual effect will be presented in the paper.

[1]National Science Foundation, grants CMMI 0826067 and CMMI 0825832.

8:52AM MG.00005 Particle dynamics and rheology of single-wall-carbon-nanotube suspensions under shear and electric fields , JERRY SHAN, CHEN LIN, Rutgers University — The particle orientation and electrorheology of dilute single-wall-carbon-nanotube suspensions was experimentally investigated. Ensemble-averaged nanotube orientation angles were measured with an optical polarization-modulation technique simultaneously with macroscopic electrorheological measurements. The results were compared with theoretical predictions of Mason. The time scales of the particle-orientation and electrorheological responses differ by an order of magnitude, indicating that nanotube alignment under the external electric field does not directly affect the rheology of the suspension at low nanotube concentrations. Equilibrium particle-orientation angles for various shear rates and electric fields were found to collapse when plotted against the ratio of shear-flow to electrostatic forces, as predicted by classical theory. However, there were significant discrepancies between the measured and predicted orientation angles of the nanotubes. It is shown that the discrepancy is due to both hydrodynamic and electrostatic interactions between particles. The significance of such interactions in dilute suspensions of highly anistropic particles under shear and electric fields is discussed.

9:05AM MG.00006 Flow Velocity Profile in a Nanocapillary , GUIREN WANG, CUIFANG KUANG, WEI ZHAO, University of South Carolina — Although there has been considerable research performed in the area of nanofluidics that is focused on the study of the velocity, only theoretical or simulation results exist. The reason for the lack of experimental evidence could be because that it is very difficulty to measure the flow velocity in nanochannel, where the transverse dimension is between 1–500 nm. We have developed a far field nanoscopic optical velocimetry that can potentially measure the velocity in nanofluidics with nanoscale spatial resolution. It is based on laser induced fluorescence photobleaching and stimulated emission depletion. In this presentation, we report our initial experimental data using water solution in a quartz nanocapillary with an inner diameter of 360 nm. The spatial resolution is better than 70 nm. First, no clear slip-flow is observed. This could probably be due to that our measurement is not accurate enough within 35 nm from the wall. Second, compared with the conventional Hagen-Poiseuille equation, the velocity profile is not purely parabolic. Instead, there seems to be a region, where the velocity profile is not smooth along radial direction and the corresponding velocity gradient is very small. Near the axial region, the velocity gradient is increased again.

9:18AM MG.00007 Surface nanobubble contact angles[1] , BRAM BORKENT, Physics of Fluids, Faculty of Science and Technology, University of Twente, SISSI DE BEER, FRIEDER MUGELE, Physics of Complex Fluids, Faculty of Science and Technology, University of Twente, DETLEF LOHSE, Physics of Fluids, Faculty of Science and Technology, University of Twente — Previous AFM experiments on surface nanobubbles have suggested an anomalously large contact angle θ of the bubbles (typically $\sim 160\,^\circ$ measured through the water) and a possible size dependence $\theta(R)$. Here we determine $\theta(R)$ for nanobubbles on smooth highly orientated pyrolytic graphite (HOPG) with a variety of different cantilevers. It is found that $\theta(R)$ is constant within the experimental error, down to bubbles as small as $R = 20\,\mathrm{nm}$, and its value is equal to $119 \pm 4\,^\circ$. This result, which is the lowest contact angle for surface nanobubbles found so far, is very reproducible and independent of the cantilever type used, provided that the cantilever is clean and the HOPG surface is smooth. In contrast we find that, for a particular set of cantilevers, the surface can become relatively rough due to precipitated matter from the cantilever onto the substrate, in which case larger nanoscopic contact angles ($\sim 150\,^\circ$) show up.

[1]this work was financially supported by NanoNed (TMM.6413) & FOM.

9:31AM MG.00008 Shear rate threshold for the boundary slip in dense polymer films[1] , NIKOLAI PRIEZJEV, Michigan State University — The shear rate dependence of the slip length in thin polymer films confined between atomically flat surfaces is investigated by molecular dynamics simulations. The polymer melt is described by the bead-spring model of linear flexible chains. We found that at low shear rates the velocity profiles acquire a pronounced curvature near the wall and the absolute value of the negative slip length is approximately equal to thickness of the viscous interfacial layer. At higher shear rates, the velocity profiles become linear and the slip length increases rapidly as a function of shear rate. The gradual transition from no-slip to steady-state slip flow is associated with faster relaxation of the polymer chains near the wall evaluated from decay of the time autocorrelation function of the first normal mode. We also show that at high melt densities the friction coefficient at the interface between the polymer melt and the solid wall follows power law decay as a function of the slip velocity. At large slip velocities the friction coefficient is determined by the product of the surface induced peak in the structure factor, temperature and the contact density of the first fluid layer near the solid wall. (Reference cond-mat/0906.2771).

[1]ACS Petroleum Research Fund

Tuesday, November 24, 2009 8:00AM - 9:57AM –
Session MH Drops VII 101H

8:00AM MH.00001 Deformation of a hydrophobic ferrofluid droplet suspended in a viscous medium under uniform magnetic fields , YURIKO RENARDY, SHAHRIAR AFKHAMI, MICHAEL RENARDY, JUDY RIFFLE, Virginia Tech, ANNETTE TYLER, TIM ST. PIERRE, ROBERT WOODWARD, The University of Western Australia — The effect of applied magnetic fields on the deformation of a biocompatible hydrophobic ferrofluid drop suspended in a viscous medium is investigated numerically and compared with experimental data. At high magnetic fields, experimental drop shapes deviate from numerical results when a constant surface tension value is used. One hypothesis for the difference is the dependence of interfacial tension on the magnetic field in the experimental data. This idea is investigated computationally by varying the interfacial tension as a function of the applied magnetic field, and by comparing the drop shapes with experimental data until matched.

8:13AM MH.00002 Motion and deformation of a hydrophobic ferrofluid droplet in a viscous medium under non-uniform magnetic fields , SHAHRIAR AFKHAMI, YURIKO RENARDY, MICHAEL RENARDY, Virginia Tech, TIM ST. PIERRE, The University of Western Australia, JUDY S. RIFFLE, Virginia Tech — Recent interest in ferrofluids has been motived by biomedical and pharmaceutical applications. We numerically investigate dynamics of a hydrophobic ferrofluid droplet placed in a viscous medium under non-uniform magnetic fields. The drop deforms in response to the applied non-uniform magnetic field, and this in turn affects the motion of the ferrofluid droplet. Droplet properties such as interfacial tension, viscosity, magnetic permeabilty as well as its size and shape can be optimized for the efficient manipulation of the motion of drop through a viscous medium. By computationally measuring the migration velocity of ferrofluid droplets, it is possible to determine the ferrofluid "magnetophoretic mobility", which is an important characteristic for the feasibility of biomedical applications.

8:26AM MH.00003 Infrared visualization of flow motion inside a sessile drop under evaporation , DAVID BRUTIN, FABRICE RIGOLLET, CHRISTOPHE LE NILIOT, Aix-Marseille University - IUSTI, TCM TEAM — Drop evaporation is a simple phenomena but still unclear concerning the evaporation mechanisms. A common agreement of the scientific community based on experimental and numerical work evidence that most of the evaporation occurs at the triple line; however, the rate of evaporation is still predicted empirically due to the lack of knowledge on the convection which develops inside the drops under evaporation. The evaporation of sessile drops is more complicated than it appears due to the conduction coupling with the heating substrate, the convection and conduction inside the drop and the convection and diffusion with the vapour phase. The coupling of heat transfer in the three phases induces complicated cases to solve even for numerical simulations. In the present work, we present recent experimental results obtained using infrared thermography (infrared camera coupled with a microscopic lens of 10 μm of resolution) to visualize flow motion inside sessile drops under evaporation. The wavelength range (3-5 μm) is adapted to the observed fluid (99.9% ethanol) since it is semi-transparent in that wavelength range. It is then possible to observe qualitatively inside the drop the convection cells appearance, evolution and disappearance. The relation between the convection cells and the heat transfer from the substrate to the drop is evidenced is this work.

8:39AM MH.00004 Small-amplitude oscillations of a drop pinned at an azimuth , DORAISWAMI RAMKRISHNA, SANTHOSH RAMALINGAM, OSMAN BASARAN, Purdue University, West Lafayette — Natural modes of oscillation of an inviscid drop undergoing irrotational flow and constrained by a ring of negligible thickness at an azimuthal angle with respect to the center of the droplet are studied analytically and numerically. Similar to linear oscillations of a free drop first studied by Rayleigh, the analytical formulation of the oscillations of the constrained drop results in an eigenvalue problem but with one additional boundary condition, i.e. that accounting for zero perturbation along the contact point. A minimization method that converts the eigenvalue problem into a constrained optimization problem is used to solve for the eigenvalues of various modes of oscillation and the corresponding mode shapes. An extension of Green's function method used to analyze oscillations of a drop in contact with a spherical bowl [M. Strani and F. Sabetta, *J. Fluid Mech.* **141**, 233 (1984)] is also employed to study the problem at hand. Results obtained from these two approaches are compared to those reported by Bostwick and Steen [*Phys. Fluids* **21**, 032108 (2009)] and ones obtained from new simulations using the Galerkin/finite element method.

8:52AM MH.00005 Trajectories of a pair of drops in steady shear at finite inertia: effects of viscosity ratio and positioning , RAJESH SINGH, KAUSIK SARKAR, University of Delaware — Two drops driven towards each other by shear, experience a new trajectory at finite inertia. Unlike in Stokes flow, where drops always slide past each other, here they turn back in a reversed trajectory due to a zone of reversed streamlines around the drops at finite inertia. The trajectory type depends on initial offset, Reynolds number, capillary number, as well as viscosity ratio. We investigate the transition from one type of trajectory to the other, and delineate the different zones in the parameter space. Drops pass each other at low and high capillary numbers, but reverse their motion at intermediate capillary numbers because of the increased drop inclination in the flow direction at increased capillary number. Above a critical viscosity ratio, drop trajectory transitions from reversed to passing. The critical viscosity ratio increases with Reynolds number at small capillary numbers, but shows a nonmonotonic behavior at higher capillary numbers. Increasing initial offset in the vorticity direction also leads to a transition from passing to reversed trajectory.

9:05AM MH.00006 Calculation of Drag and Lift for a Deformable Droplet in a Shear Flow , YOUNGHO SUH, CHANGHOON LEE, Yonsei University — Significant efforts have recently been made to numerically investigate the drag and lift forces of a liquid droplet in a shear flow. The droplet deformation induced by interplay between surface tension and shear stress on the surface can affect the droplet behavior. In most previous studies, however, only the spherical droplet without deformation has been considered in the shear flow, and thus understanding of effects of droplet deformation on drag and lift characteristics lacks. In this study, the droplet behavior in the linear shear flow is numerically investigated 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. Computations were carried out to investigate the deformation behavior of droplet caused by the linear shear flow with various shear rate, droplet size and flow velocity. Also, the effects of deformation of droplet on drag and lift forces acting on droplet are presented. In the computation, to determine the acting force on a droplet in shear flow field, the feedback forces which can maintain droplet position were adopted for efficient handling of deformation.

9:18AM MH.00007 Numercial study of deformable droplet-pair subjected to sudden acceleration by an external flow , SHAOPING QUAN, JING LOU, Institute of High Performance Computing — Numerical simulations of two deformable drops placed in tandem subjected to a sudden acceleration by a gaseous flow are performed by solving the integral form of the full Navier-Stokes equations using a finite volume/moving mesh interface tracking method. The interface is zero thickness and moves in a Lagrangian fashion. The unsteady interaction between the droplet-pair is studied by varying the minimum initial distance between the two droplets. The interaction on the shapes, the deformation factors, the drag coefficients, and the fluid fields are examined and compared to the single droplet. A mushroom shape is formed with a dimpled cap of the upstream drop and a bell shape of the downstream one for high Weber numbers and small initial separation distances. The drag coefficient of the downstream droplet is dramatically reduced, especially for the large Weber numbers, while drag force of the upstream drop is slightly decreased. It is found that when the spacing is less than zero, there is a sudden increase of drag coefficient for the downstream drop, while a sharp reduction for the upstream drop. For smaller Weber numbers, the two droplets experience oscillations, however, the oscillation modes are not the same. The two drops might coalescence for small separation distances.

9:31AM MH.00008 Radiation torque on an absorptive spherical drop centered on an acoustic helicoidal Bessel beam , LIKUN ZHANG, PHILIP L. MARSTON, Washington State Univ. Physics and Astronomy Dept. — Circularly polarized electromagnetic waves carry axial angular momentum and analysis shows that the axial radiation torque on an illuminated sphere is proportional to the power absorbed by the sphere [1]. Helicoidal acoustic beams also carry axial angular momentum and absorption of such a beam should also produce an axial radiation torque [2]. In the present work the acoustic radiation torque on solid spheres and spherical drops centered on acoustic helicoidal Bessel beams is examined. The torque is predicted to be proportional to the ratio of the absorbed power to the acoustic frequency. Depending on the beam helicity, the torque is parallel or anti-parallel to the beam axis. The analysis uses a relation between the scattering and the partial wave coefficients for a sphere in a helicoidal Bessel beam. Calculations suggest that beams with a low topological charge are more efficient for generating torques on solid spheres.

[1] P. L. Marston and J. H. Crichton, Phys. Rev. A. 30, 2508-2516 (1984).
[2] B. T. Hefner and P. L. Marston, J. Acoust. Soc. Am. 106, 3313-3316 (1999).

9:44AM MH.00009 Radiation force on drops and bubbles in acoustic Bessel beams modeled using finite elements[1] , PHILIP L. MARSTON, DAVID B. THIESSEN, LIKUN ZHANG, Washington State Univ. Physics and Astronomy Dept. — Analysis of the scattering of sound by spheres centered on ordinary and helicoidal (higher-order) Bessel beams makes it possible to evaluate the acoustic radiation force on idealized drops and bubbles centered on the beam [1]. For potential applications it would be necessary to know if a small transverse displacement of the sphere from the beam's axis causes a radiation force that pushes the sphere toward (or away from) the axis of the beam. We applied 3D-finite elements to that problem. To trust FEM calculations of the radiation force with helicoidal beams it was first necessary to verify that analytical values for the axial force are recovered in the on-axis helicoidal case since only the zero-order beam had been previously studied with FEM. Cases have been identified where the force pushes a slightly off-set drop or bubble toward the axis. For some cases the effective potential method of Gorkov may be used to predict the transverse stability of small spheres.

[1] P. L. Marston, J. Acoust. Soc. Am. 125, 3539-3545 (2009).

[1]Supported in part by ONR and NASA.

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MJ Surface Tension Effects II 101I

8:00AM MJ.00001 Friction Reduction in Superhydrophobic Microchannels , TAE JIN KIM, CARLOS H. HIDROVO, The University of Texas at Austin — Superhydrophobic surfaces are surfaces with fluid contact angles larger than 150°. Superhydrophobicity can be achieved by means of surface texturing through either a Wenzel or Cassie state. It is widely known, however, that drag reduction is closely related to Cassie state surfaces with low degrees of adhesion and several studies have been widely conducted on the topic. In this research we investigate the effects of surface texturing on superhydrophobic microchannels. Both PDMS and silicon based samples were fabricated and used to experimentally characterize the effects that microtexturing geometry has on the friction reduction behavior. We developed a layered-two-fluid system model to simulate the slip velocity condition and approximate the drag reducing behavior of the microtexturing. A surface energy formulation was also introduced to account for the effects that pressure has on the transition from a Cassie (non-wetting) to Wenzel (wetting) state of the microtexturing. Pressure effects in the wetting of the microtexturing are essential since the flow is driven by a pressure gradient. Experimental results on both friction reduction and pressure induced microtexturing wetting were compared against the models.

8:13AM MJ.00002 Hydraulic Jumps on Superhydrophobic Surfaces Exhibiting Ribs and Cavities , MICHAEL JOHNSON, BENTON RUSSELL, DANIEL MAYNES, APS, BRENT WEBB — We report experimental results characterizing the dynamics of a liquid jet impinging normally on superhydrophobic surfaces spanning the Weber number (based on the jet velocity and diameter) range from 100 to 1400. The superhydrophobic surfaces are fabricated with both silicon and PDMS surfaces and exhibit micro-ribs and cavities coated with a hydrophobic coating. In general, the hydraulic jump exhibits an elliptical shape with the major axis being aligned parallel to the ribs, concomitant with the frictional resistance being smaller in the parallel direction than in the transverse direction. When the water depth downstream of the jump was imposed at a predetermined value, the major and minor axis of the jump increased with decreasing water depth, following classical hydraulic jump behavior. When no water depth was imposed, however, the total projected area of the ellipse exhibited a nearly linear dependence on the jet Weber number, and was nominally invariant with varying hydrophobicity and relative size of the ribs and cavities. For this scenario the Weber number (based on the local radial velocity and water depth prior to the jump) was of order unity at the jump location. The results also reveal that for increasing relative size of the cavities, the ratio of the ellipse axis (major-to-minor) increases.

8:26AM MJ.00003 Sinking of a sphere into viscous liquid , DUCK-GYU LEE, HO-YOUNG KIM, Seoul National University — A dense solid sphere gently released on an air-liquid interface slowly sinks into the liquid due to gravity while the motion is resisted by liquid viscosity and interfacial tension. We study the sinking velocity of the sphere both theoretically and experimentally. The viscous drag force on the sphere is determined by solving the Stokes equation. To find the retarding force due to interfacial tension, we obtain the meniscus profile by solving the dynamic boundary condition that relates the jump of normal stresses across the air-liquid interface to the surface tension. The predicted sinking velocity, a function of the sphere density and radius, liquid density, viscosity and surface tension, and the dynamic contact angle, is in good agreement with the experimental measurements. The current work expands our knowledge of the sinking process of small spheres which mainly concerned the sinking of completely immersed spheres so far.

8:39AM MJ.00004 Capillary aggregates of floating particles as an attractive granular media , MICHAEL BERHANU, ARSHAD KUDROLLI, Clark University — Aggregation of cohesive particles floating in a medium is a very broad physical phenomena occurring in colloidal systems, soot particles, and intergalactic dust under gravitation. We investigate the geometrically constrained dynamics of aggregation with new experiments using floating spheres (3 mm) at the air-liquid interface. A short range attractive force can be induced by the combination of buoyancy and capillarity to create self-assembled particle structures which can be tracked by imaging. First, the particles are placed randomly at the interface, and then aggregation is induced by smoothly decreasing the area of the interface which causes the particles to come within the attractive force range caused by capillarity. We study the aggregation phenomena due to the fusion of initial small clusters in one large cluster. Once it is formed, we are interested by its mechanical properties, and the question of the cluster rigidity is discussed. Finally we study the structural properties of aggregates. Thus we can exhibit significant differences with a classical two dimensional granular media due to attraction between particles.

8:52AM MJ.00005 Forced Spreading and Coalescence of Viscous Drops , SHELLEY ANNA, PILGYU KANG, Carnegie Mellon University, SHAHAB SHOJAEI-ZADEH, Benjamin Levich Institute, CHRISTINE APPLEBY, Carnegie Mellon University — This study investigates the dynamics of spreading and coalescence of sessile droplets on a surface, a process important in applications such as inkjet printing, spray coating, and flooding of fuel cells. We use a simple microfluidic device to control the spreading and merging processes. Droplet shape, diameter and maximum height are monitored as functions of time. We compare the dynamics with existing scaling models modified to incorporate time dependent volume, and we extend these models to describe the scaling behavior of the liquid bridge growing between merging droplets on a surface. The experiments agree well with the expected scaling incorporating capillary, gravity, and viscous forces.

9:05AM MJ.00006 Computational Study of Moving Striple Lines , NEEHARIKA ANANTHARAJU, MAHESH PANCHAGNULA, SRIKANTH VEDANTAM, Tennessee Technological University — Wetting of chemically heterogeneous surfaces is modeled using phase field theory. This model studies the one-dimensional kinetic processes involved in wetting a substrate similar to a Wilhelmy technique. The experimental technique provides a complete validation due to its capability to capture the triple line kinetics in addition to measuring the contact angles during the advancing and receding processes. A chemically heterogeneous surface is said to be composed of a predetermined arrangement of two materials. The novelty in the current approach lies in the fact that each of the component materials is constitutively allowed to exhibit hysteresis. We investigate the local shape of the triple line which plays an important role in determining the macroscopic contact angle due to its ability to be pinned at various defect locations on real surfaces. We also demonstrate that the shape of the advancing and receding triple line is sensitive to the specific arrangement of the two materials.

9:18AM MJ.00007 A simplified hydrodynamic study of painting , JUNGCHUL KIM, HO-YOUNG KIM, Seoul National University — The use of a paintbrush by mankind is known to have started 2000 BC in the Chinese ancient empire, Jin. Despite the long history of painting, attempts to physically and mathematically understand the process of painting seem sparse so far. Here we consider how paint is applied on a solid surface by studying the behavior of a viscous drop sheared between moving plates simulating a canvas and a paintbrush. Dimensional analysis reveals that the behavior is determined by the Capillary number (a ratio of the viscous force to the surface tension force), the receding contact angle and the drop aspect ratio. We experimentally find three distinct drop behaviors, intact dragging, dripping and spreading, and construct a regime map using the foregoing dimensionless parameters. We also give scaling laws to determine the boundaries on the regime map, which agree well with experiment.

9:31AM MJ.00008 Combined Gravitational and Thermocapillary Interactions of Spherical Drops with Incompressible Surfactant , MICHAEL ROTHER, University of Minnesota Duluth — Collision efficiencies are calculated by a trajectory analysis for two contaminated spherical drops under the combined influence of buoyancy and a constant temperature gradient at low Reynolds number and with negligible thermal convection in the limit of nearly uniform surfactant coverage. As in the case of clean drops, a region in the parameter space exists where collisions are forbidden when the driving forces are opposed. However, because of the increased effect of thermocapillary repulsion when surfactant is present, coalescence can be inhibited even when the driving forces are aligned in the same direction. In addition to trajectories leading to coalescence and separation of the drops, closed trajectories are also observed. At parameter values where the asymmetric mobility function is zero, retrograde motion can occur, where the angle between vertical and the drops' line of centers decreases as the drops come into contact. This retrograde motion requires alteration to the closed form expression for the collision efficiency. The effect of incompressible surfactant on dilute dispersions of two physical systems is also considered.

9:44AM MJ.00009 Surfactant Effects on Drops Rising in Tubes , YUANYUAN CUI, NIVEDITA GUPTA, University of New Hampshire — We present our numerical results for the motion of viscous drops rising in cylindrical capillaries at finite Reynolds numbers. A hybrid volume-of-fluid method with a front-tracking scheme is implemented to explore the effects of Bond number, Weber number, and viscosity of the drop phase on the steady shapes and terminal velocities of drops. As the Bond number increases, drops become more deformed. At higher Weber numbers, a reentrant cavity is found at the rear of the drop. Increasing the viscosity of the drop phase reduces the terminal velocity since the resistance of the drop to the ambient fluid increases. We also present the results of adding soluble surfactants in the adsorption-desorption limit in the two-phase system to study the effects of surfactant concentration on the drop motion. The surfactants are modeled using a Langmuir adsorption framework. The non-uniform distribution of surfactants along the fluid interface generates Marangoni stresses that retards the motion of drops and depends on the initial coverage of surfactants. We investigate the effect of mass transfer kinetics on drop deformation and mobility.

9:57AM MJ.00010 Surfactant-induced migration of a drop in Stokes flow , JAMES HANNA, PETIA VLAHOVSKA, Thayer School of Engineering, Dartmouth College — In Stokes flows, symmetry considerations dictate that a neutrally-buoyant spherical particle will not migrate laterally with respect to the local flow direction. We show that a loss of symmetry due to flow-induced surfactant redistribution leads to cross-stream drift of a spherical drop in Poiseuille flow. We derive analytical expressions for the migration velocity in the limit of small non-uniformities in the surfactant distribution, corresponding to weak-flow conditions or a high-viscosity drop. The analysis predicts that the direction of migration is always towards the flow centerline.

Tuesday, November 24, 2009 8:00AM - 10:10AM –

Session MK Particle Laden Flows III 101J

8:00AM MK.00001 Analysis of dense particulate flow dynamics using a Euler-Lagrange approach , OLIVIER DESJARDINS, Mechanical Engineering, University of Colorado at Boulder, PERRINE PEPIOT, National Renewable Energy Laboratory — Thermochemical conversion of biomass to biofuels relies heavily on dense particulate flows to enhance heat and mass transfers. While CFD tools can provide very valuable insights on reactor design and optimization, accurate simulations of these flows remain extremely challenging due to the complex coupling between the gas and solid phases. In this work, Lagrangian particle tracking has been implemented in the arbitrarily high order parallel LES/DNS code NGA [Desjardins et al., JCP, 2008]. Collisions are handled using a soft-sphere model, while a combined least squares/mollification approach is adopted to accurately transfer data between the Lagrangian particles and the Eulerian gas phase mesh, regardless of the particle diameter to mesh size ratio. The energy conservation properties of the numerical scheme are assessed and a detailed statistical analysis of the dynamics of a periodic fluidized bed with a uniform velocity inlet is conducted.

8:13AM MK.00002 Rheology, Microstructure and Migration in Brownian Colloidal Suspensions , WENXIAO PAN, BRUCE CASWELL, GEORGE KARNIADAKIS, Brown University — We demonstrate that suspended spherical colloidal particles can be effectively modelled as single dissipative particle dynamics (DPD) particles provided that the conservative repulsive force is appropriately chosen. The suspension model is further improved with a new formulation, which augments standard DPD with non-central dissipative shear forces between particles while preserving angular momentum. Using the new DPD formulation we investigate the rheology, microstructure and shear-induced migration of a monodisperse suspension of colloidal particles in plane shear flows (Couette and Poiseuille). Our simulations yield relative viscosity versus volume fraction predictions in good agreement with both experimental data and empirical correlations. We also compute the shear-dependent viscosity and the first and second normal- stress differences and coefficients in both Couette and Poiseuille flow. Simulations near the close packing volume- fraction (64%) at low shear rates demonstrate a transition to flow-induced string-like structures of colloidal particles simultaneously with a transition to a non-linear Couette velocity profile. Migration effects simulated in Poiseuille flow compare well with experiments and model predictions. Overall, the new method agrees very well with the Stokesian Dynamics method but it seems to have lower computational complexity and is applicable to general complex fluids systems.

8:26AM MK.00003 Discrete Element Modeling of Particle Transport by an Electric Curtain[1] , JEFFREY MARSHALL, University of Vermont, GUANQING LIU, Tsinghua University — Particle transport by a planar electric curtain is studied with a discrete element method (DEM). The electric curtain is consists of four sets of parallel stripe electrodes embedded in an insulating material. The electrodes are connected to a four-phase square-wave AC source and produce a traveling wave above the curtain plate. Current work focuses on different modes of particle transport and associated operating conditions, which include particle relevant properties (size, charge, adhesive surface energy, air drag) and device relevant parameters (amplitude, frequency of applied AC voltage). Both Coulomb and dielectrophoretic forces on particles are considered. The electric field produced by the curtain is solved by a two-dimensional boundary element method (BEM). The computation is validated by comparison with experimental data for particle transport speed. A series of simulations are conducted to investigate the influence of different factors, demonstrating different modes of particle transport. The effect of air flow over the curtain on transport of levitated particles is examined. The particle motion is found to exhibit a wealth of interesting phenomena due to the complex and combined affects of different physical parameters and processes.

[1]Project sponsored by NASA and the China Scholarship Council.

8:39AM MK.00004 Modeling Visco-elastic Particle Collision in Coupled Direct Numerical/Discrete Particle Simulations , JULIAN SIMEONOV, JOSEPH CALANTONI, Marine Geosciences Division, Naval Research Laboratory, Code 7440.3, Stennis Space Center, MS 39529, USA — We study particle collision with coupled Direct Numerical Simulations (DNS) and Discrete Particle Simulations (DPS) where the flow and the particle evolution are determined from the Navier-Stokes and Newton's equations of motion, respectively. The hydrodynamic force on a particle is obtained by integrating the resolved pressure and viscous stress on the particle surface, and the normal and tangential particle- contact forces are modeled with springs and friction. Resolving visco-elastic collisions in DNS/DPS requires integrating the fluid equations at time steps of a few microseconds that are prohibitively small for modeling large turbulent systems. We develop a robust collision scheme for coupled DNS/DPS where the fluid integration time step is much larger than the particle time step. The scheme is based on two- particle DNS/DPS with matching fluid and particle time steps. The visco-elastic restitution coefficient obtained in the two-particle simulations and its dependence on the collisional Stokes number is compared to experimental results.

8:52AM MK.00005 Coarse grid simulation of dense fluidized bed: a dynamic subgrid drag model , JEAN-FRANCOIS PARMENTIER, OLIVIER SIMONIN, Institut National Polytechnique de Toulouse, OLIVIER DELSART, TOTAL CReG — Unresolved structures in 3D unsteady simulation of gas-particle fluidized bed using the two-fluid model approach can have a drastic influence on the flow dynamic. In particular, dense fluidized bed expansion may be widely overestimated. Such an effect can be inputed to the coarse modeling of the resolved drag term in the momentum equations. Filtered two-fluid model formalism highlights the need to account for a drift velocity due to the subgrid correlation between the fluid velocity and particle concentration fields. A priori analysis were performed from highly resolved simulations of dense fluidized beds. Results gathered provide us constitutive relationship to close this drift velocity and we proposed a dynamic model to predict the filtered drag force. This model was a-posteriori tested on "coarse-grid" fluidized bed simulations. Simulation results compared fairly well to fully resolved simulations and experimental bed expansions.

9:05AM MK.00006 Partitioning of particle temperature in gas-solid turbulent flows , ENRICA MASI, OLIVIER SIMONIN, PASCAL FEDE, IMFT — In this work we characterize the particle temperature distribution of inertial particles interacting with a turbulent non-isothermal flow. In the general frame of the mesoscopic Eulerian approach developed by Fevrier et al. [*J. Fluid Mech.*, 533, 1-46] for the particle velocity distribution, we show that the instantaneous particle temperature can be partitioned in two different contributions. The first contribution is a continuous temperature field shared by all the particles and taking into account two-point correlations. The second is a random spatially uncorrelated contribution, characterized in terms of Eulerian fields of particle moments. This conclusion is pointed out through statistical one-point and two-point correlation analysis. Statistics are measured from numerical simulations of particles suspended into a homogenous, isotropic, stationary non- isothermal turbulence where discrete particle paths are computed by Lagrangian tracking. The goal of this work is to improve the comprehension of mechanisms of heat transfer and transport for the development of unsteady Eulerian modelling approaches for reactive two-phase flows. The local instantaneous Eulerian equations for the mesoscopic temperature and the random uncorrelated heat flux are derived. We also point out that temperature distribution is related to the both dynamical and thermal particle response times.

9:18AM MK.00007 Particle motion in a turbulent serpentine channel , XIN HUANG, PAUL DURBIN, Iowa State University — Particle motion in serpentine passage was studied using direct numerical simulation coupled with Lagrangian particle tracking. In a straight, turbulent channel, eddies are responsible for particle deposition in the near-wall region. In a serpentine channel, particle inertia may be the dominant cause of high concentration near the outer bend in the curved section. However, turbulent convection is still important. This depends on the relative magnitude of particle Stokes number and turbulent time-scale. Particles with small Stokes number form a plume that leaves the inner bend and high concentration layers near both walls in the straight section before entering the bend. The high concentration layers remain thin, reminiscent of plane channel flow. Significant deposition of heavier particles in the straight section is only found on the side extended from the outer bend. There are two plumes at the entrance of bend because of the larger particle inertial and thicker layer of high concentration. Particle deposition in the curved section is also affected by particle-wall collision and simulation results of particulate impact and erosion will be presented.

9:31AM MK.00008 Incipient motion of spherical particles induced by a vortex ring disturbance , MARIA-LAURA BENINATI, Bucknell University, MICHAEL MCERLEAN, MICHAEL KRANE, ARNOLD FONTAINE, Pennsylvania State University — Experiments to characterize the ability of a vortical disturbance to induce incipent motion of a stationary particle resting on a horizontal planar surface are described. The ultimate goal of this study is to assess the role of turbulent boundary layer coherent structures in particle suspension. In this study, a vortex ring plays the role of a model flow disturbance because its compact, three-dimensional structure and speed are easily controlled and characterized. The vortex rings are generated by injecting a slug of fluid at a specified height above the ground plane. The vortex ring and its motion are described in terms of the convection speed, size, and circulation, using a combination of high-speed video and DPIV. The flow disturbance at the wall is characterized using wall pressure measurements during the passage of the vortex ring. These measurements are used to determine the relationship between the measures of the vortex ring disturbance and the resulting pressure fluctuation at the wall. Finally the ability of a vortex ring of known strength and convection path to induce motion to a single particle placed on the ground plane is assessed using high-speed video.

9:44AM MK.00009 Optical measurements of jet gas and bed particle velocity distributions in a 2D bubbling fluidized bed[1] , ALEXANDER MYCHKOVSKY, STEVEN CECCIO, University of Michigan — A laser Doppler velocimetry (LDV) measurement technique has been developed to obtain spatially and temporally resolved measurements of jet gas and bed particle velocities in a 2D bubbling fluidized bed. The LDV system was configured to eliminate spurious optical intensity fluctuations, which can contaminate velocity measurements in optically dense flows. The jet gas was seeded with ice crystals, which were formed by rapidly condensing and freezing the moisture in the jet air just prior to injection. LDV bursts from the bed particles and gas tracer ice crystals were simultaneously recorded to obtain the particulate and gas phase velocities at a given location within the jet plume in a non-intrusive manner.

[1]Department of Energy

9:57AM MK.00010 Particle Capture by a Conducting Cylinder in an Electrostatic Field using a Discrete Element Method[1] , GUANQING LIU, SHUIQING LI, Tsinghua University, JEFFREY MARSHALL, University of Vermont — Particle capture by a conducting cylinder is studied with a discrete element method (DEM) developed for particle transport with electrostatic effects. A charged cylinder is placed in a uniform electric field. Uncharged particles are advected toward the cylinder by air flow. Electric field generation by the cylinder is resolved by a boundary element method (BEM), which accounts for effect of polarized particles on the cylinder induced surface charge. Conventional BEM exhibits errors when near-surface particles are smaller than BEM panels. An algorithm using approximate particle images and local panel subdivision is introduced to improve computational accuracy. Particle-particle electrostatic interaction is accelerated using a fast multipole expansion method. The simulation shows that particles captured by the cylinder form straight chains oriented nearly perpendicular to the cylinder surface. Varying the cylinder voltage with fixed uniform electric field strength leads to different particle deposition characteristics. Particles deposit on only one side of the cylinder at low voltages, but at higher voltages particles deposit all over the cylinder. Predicted particle capture efficiency compares well with experimental data.

[1]Supported by NASA and China Scholarship Council.

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session ML General Fluid Mechanics I 200A

8:00AM ML.00001 PIV Analysis of Prandtl's Flow Visualization Movies , CHRISTIAN WILLERT, JUERGEN KOMPENHANS, German Aerospace Center, MORTEZA GHARIB, Caltech — Probably some of the oldest time-resolved PIV image sequences still available today were recorded on film during the late 1920's and early 1930's by Prandtl and his colleagues Tietjens and Müller using free surface water flumes at the Kaiser-Wilhelm-Institute for Fluid Research (Göttingen), now the Max Planck Institute for Self Organisation. Recorded at 20 frames per second the films visualize the process of unsteady flow separation and vortex generation on surface piercing objects such as airfoils or cylinders. The visualization was achieved by means of small particles (aluminum powder, ferrous mica or lycopodium powder) scattered on the water surface. Illumination from above resulted in high contrast images of the random particle distribution that are very well suited for PIV analysis. Modern PIV software is used to process digitized versions of the films, made available by the "Institut für wissenschaftlichen Film" (www.iwf.de) in Göttingen. In addition to the surface flow field the time evolving vorticity field and other quantities can now be visualized which by itself shows the importance of carefully documenting and archiving valuable data.

8:13AM ML.00002 POD Analysis of Jet-Plume/Afterbody-Wake Interaction , NATHAN E. MURRAY, JOHN M. SEINER, BERNARD J. JANSEN, LICHUAN GUI, SHUAN SOCKWELL, MATTHEW JOACHIM, University of Mississippi — The understanding of the flow physics in the base region of a powered rocket is one of the keys to designing the next generation of reusable launchers. The base flow features affect the aerodynamics and the heat loading at the base of the vehicle. Recent efforts at the National Center for Physical Acoustics at the University of Mississippi have refurbished two models for studying jet-plume/afterbody-wake interactions in the NCPA's 1-foot Tri-Sonic Wind Tunnel Facility. Both models have a 2.5 inch outer diameter with a nominally 0.5 inch diameter centered exhaust nozzle. One of the models is capable of being powered with gaseous H_2 and O_2 to study the base flow in a fully combusting senario. The second model uses hi-pressure air to drive the exhaust providing an unheated representative flow field. This unheated model was used to acquire PIV data of the base flow. Subsequently, a POD analysis was performed to provide a first look at the large-scale structures present for the interaction between an axisymmetric jet and an axisymmetric afterbody wake. PIV and Schlieren data are presented for a single jet-exhaust to free-stream flow velocity along with the POD analysis of the base flow field.

8:26AM ML.00003 Interaction of a wall jet with wake behind a bluff body in an open channel , ARINDAM SINGHA, RAM BALACHANDAR, University of Windsor — Junction flow around a bluff body mounted in bed has been the subject of investigation for decades because of the unique phenomenon associated with the generation and dynamics of the horseshoe vortices. However, a more interesting flow could be the case where a small bleed flow in the form of a wall jet is allowed to pass beneath the bluff body and let interact with the horseshoe vortex. If this flow is strong, it can completely detoriates the effect of the horseshoe vortex and may result into a complex three-dimensional flow. To explore the characteristics of this kind of flow a sharp-edged bluff body was mounted in an open channel flow of nominal flow depth of 100 mm. Three flow configurations were examined: a) the body was mounted firmly on the floor, b) the body was lifted 5 mm from the bed and c) the body was lifted 10 mm from the bed. Particle image velocimetry measurement was performed at three horizontal planes, at $y/H = 0.10$, 0.50 and 0.80, respectively, where H is the nominal depth of flow. Time-averaged flow parameters were examined at selected streamwise locations at different vertical elevations to examine the effect of wall jet to change the characteristics of the wake.

8:39AM ML.00004 Passive Scalar Transport in Pipe Flows , KEITH MERTENS, University of North Carolina, UNC RTG Joint Fluids Lab, ROBERTO CAMASSA, RICHARD MCLAUGHLIN, MATTHEW MOORE, MATT HERNANDEZ, University of North Carolina, UNC RTG Joint Fluids Lab — The problem of passive scalar transport in pipe flows has a long standing history. Recent work has been re-examining the concentration evolution during the initial transient timescale before G.I. Taylor's 1953 theory becomes applicable. Using high resolution digital photography we experimentally investigate this transient concentration evolution in laminar pipe flow. Gravitational effects associated with non-homogeneous densities induced by the passive scalar are shown to play a role, especially at short timescales, and need to be carefully mitigated through density matching. In density matched experiments, we observe anomalous behavior in the form of the development of non-zero skewness and identify the relevant timescales of these anomalies. Comparisons with theoretical predictions, including recent advancements based on a mathematically rigorous stochastic differential equations approach, will be presented.

8:52AM ML.00005 Experimental and Numerical Investigation of Supercritical Carbon dioxide compact heat exchanger , ROMA FATIMA, Department of Mechanical Engineering, Texas A&M University, ALAN KURIZENGA, MARK ANDERSON, University of Wisconsin-Madison, DEVESH RANJAN, Department of Mechanical Engineering, Texas A&M University — The use of super-critical carbon dioxide is gaining importance because of its use in Brayton cycles, to increase the cycle efficiency and reduce the initial capital investment, for high temperature energy conversion system. In order to reduce the capital cost, one improvement which was thought, is the use of compact, highly efficient, diffusion bonded heat exchangers for the regenerators. In this presentation we will focus on the experimental measurements of heat transfer and pressure drop characteristics within mini-channels. Two test section channel geometries were studied: a straight channel and a zigzag channel. Both configurations are 0.5m in length and constructed out of 316 stainless steel with a series of nine parallel 1.9mm semi-circular channels. The zigzag configuration has an angle of 115 degrees with an effective length of ∼0.58m. Heat transfer measurements were conducted for varying ranges of inlet temperatures, pressures, and mass flow rates. Numerical simulations have been performed using Fluent 12.0 to complement our experimental program. This is an ongoing program and we will be showing our recent progress we have made in last six months.

9:05AM ML.00006 Generation of Vertical Density Layering by Moving Boundaries[1] , MATTHEW MOORE, ROBERTO CAMASSA, RICHARD MCLAUGHLIN, UNC RTG Fluids Group, UNC RTG FLUIDS GROUP TEAM, JOINT FLUIDS LAB TEAM, CAROLINA CENTER FOR INTERDISCIPLINARY APPLIED MATHEMATICS TEAM — A vertically moving boundary in a stratified fluid can create and maintain vertical layering of density (i.e., a horizontal density gradient). We theoretically and experimentally investigate the scenario in which the gradient of density is sharp and localized, as well as non-diffusive. A careful experimental study of a vertically towed fiber through a sharp, stably stratified, highly viscous fluid will be presented. We observe a vertical column of dense fluid entrained by the moving fiber with a well-defined diameter. We investigate various theoretical models for predicting the diameter of this column, including an approach from an unbounded fluid domain limit, a lubrication model, and finally linear stability analysis of a family of exact solutions for bounded and unbounded domains. We compare these theoretical predictions with the experimental observations.

[1]Supported by NSF ATM-0327906 and NSF DMS-0502266.

9:18AM ML.00007 Sky dancer: an intermittent system , ANNE CROS, JESSE ALEXANDER RODRÍGUEZ ROMERO, OSCAR DAMIÁN DÍAZ ANDRADE, Universidad de Guadalajara — Sky dancers attract people sight to make advertising. What is the origin of those large vertical tubes fluctuations above an air blower? This study complements the previous one [1] about the system analysis from a dynamical system point of view. As a difference from the "garden hose-instability" [2], the tube shape has got "break points". Those "break points" separate the air-filled bottom tube portion from its deflated top portion. We record the tube dynamics with a high-speed videocamera simultaneously that we measure the pressure at the air blower exit. The intermittent pressure evolution displays picks when the tube fluctuates. We compare those overpressure values with the ones that appears in a rigid tube whose exit is partially obstructed. [1] F. Castillo Flores & A. Cros "Transition to chaos of a vertical collapsible tube conveying air flow" *J. Phys.: Conf. Ser.* **166**, 012017 (2009). [2] A. S. Greenwald & J. Dungundji "Static and dynamic instabilities of a propellant line" *MIT Aeroelastic and Structures Research Lab, AFOSR Sci.* Report: AFOSR 67-1395 (1967).

9:31AM ML.00008 Simplified models of high-aspect-ratio ellipsoids under shear , FENG SHI, PETER J. MUCHA, UNC-Chapel Hill — Inspired by rheoscopic flow visualization, we extend the classic study of the motion of small ellipsoidal particles under shear, focusing on simplifications obtained by consideration of the extreme aspect ratios typical of rheoscopic particles (e.g., Kalliroscope). In particular, the long-time behavior of scalene (triaxial) elllipsoids can in some cases be well-approximated by a low-order model in the appropriate aspect ratios. We enumerate and describe the generic long-time motions of such particles in the lowest-order model. We then investigate changes induced by inclusion of the physically-appropriate first-order correction, with particular attention to a periodic wobbling motion special to scalene (cf. axisymmetric) ellipsoids.

9:44AM ML.00009 Innovative Method for Greatly Reducing Flow Resistance and Obtaining Well-Ordered Continuous Flow , WEIYI LIN — In this paper, firstly, the experiment on the flow resistance of the aerated pipe flow is introduced. And some experimental research on comparison between different volumes of air entrained is presented. Secondly, the characteristics of Gravity Pipe Flow under the action of Torricelli's Vacuum, shortly called as GPFUTV are dissertated, including creative and functional design, fundamental principle, etc. Under GPFUTV condition the water flow in the tube is full-pipe and continuous, colorless and non-aerated, high-speed and non-rotational as distinguished from laminar flow. Thirdly, an appeal in relation to the experimental research, the applied studies and basic theory research is given. For instance, the well-known Reynolds' experiment under GPFUTV condition, the potential for GPFUTV to be developed for deep seawater suction technology, seawater intake pipe of OTEC and lifting technology for deep ocean mining in Fe-Mn concretions, flow stability and flow resistance under GPFUTV condition, etc.

9:57AM ML.00010 Onsager reciprocity relations derived from Maxwell reciprocity relations , HOWARD BRENNER, MIT — Based upon recent work by the author [H. Brenner, "Bi-velocity hydrodynamics," Physica A 388 (2009) 3391-3398] devoted to the foundations of fluid mechanics and transport processes, a simple model of linear irreversible thermodynamics (LIT) is developed. On the basis of this the Onsager Reciprocal Relations (ORR) applicable to LIT are shown by purely macroscopic arguments to be a direct consequence of Maxwell's Reciprocal Relations (MRR) governing equilibrium thermodynamics. Among other things, this work answers a fundamental question raised by B. D. Coleman and C. Truesdell ["On the reciprocal relations of Onsager," J. Chem. Phys. 33 (1960) 28-31] regarding the need for a rational basis for choosing a compatible set of forces and fluxes for which the ORR are a priori valid.

Tuesday, November 24, 2009 8:00AM - 9:57AM –
Session MM Reacting Flows III 200B

8:00AM MM.00001 Effects of backround turbulence on a wrinkled flame , FRANCESCO CRETA, MOSHE MATALON, University of Illinois at Urbana-Champaign — We study the propagation of a premixed flame in the wrinkled laminar flamelet regime within the context of a hydrodynamic theory, i.e., where flame thickness is considered small compared to fluid flow characteristic length scales. The flame sheet separates burnt and unburnt zones each at a given constant density. We solve a level set equation for the flame interface propagating at a flame speed modeled by flame stretch and modulated by a Markstein length and advected by the local fluid flow. The ensuing dual density pattern is fed to the Navier-Stokes equations, which provide the updated advecting velocity field. To investigate the influence of background turbulence on flame propagation, the flame is subjected to a turbulent field parameterized by its intensity and integral scale at the flame interface. To hold these parameters constant a feedback control system is implemented acting on the inflow characteristics. The local flow field affects the wrinkling of the flame and its overall propagation speed through a purely kinematic process. The influence of the chemical and diffusive processes is studied by varying functional parameters, such as Markstein length and expansion ratio. Thus, the influence of turbulence intensity and scale on the overall flame propagation is statistically determined with the additional parametric effect of each of the functional parameters and of flame curvature and strain rate.

8:13AM MM.00002 Experimental Study of Reaction and Vortex Breakdown in a Swirl-Stabilized Combustor , CHUKWUELOKA O.U. UMEH, General Electric Aviation, ZVI RUSAK, RPI, EPHRAIM J. GUTMARK, University of Cincinnati — Non-reacting and lean reacting flow experiments are conducted in a swirl-stabilized combustor with several configurations of a TARS fuel injector. The test chamber is composed of the TARS swirler at the inlet of a straight cylindrical fuel pre-mixing section, followed by a sudden expansion and a finite-length concentric chamber, open to the atmosphere and with optical access. Non-reacting flow tests are conducted with air at 300K and 600K, while reacting flow tests use premixed air (at 600K) and gaseous propane fuel. Simultaneous PIV measurements and OH chemiluminescence is taken and used to describe the velocity field and location of vortex breakdown and reaction zones. Results show the complex dynamical interaction between the flame and breakdown zone, and the oscillations in the position of both. In the non-reacting cold flows, the breakdown zone appears near the expansion plane. In the non-reacting pre-heated flows it is pushed downstream of the expansion plane. For reacting flows with equivalence ratios near the lean blow out point, the breakdown zone is anchored near the expansion plane, while the flame oscillates inside it. At higher equivalence ratios, the flame is anchored near the expansion plane while the breakdown zone oscillates behind it. Measured swirl numbers show nice correlation with breakdown position and satisfy necessary theoretical conditions.

8:26AM MM.00003 Heat release in freely-propagating lean premixed hydrogen-methane mixtures , XINFENG GAO, MARCUS DAY, JOHN BELL, Lawrence Berkeley National Lab — Freely-propagating lean premixed hydrogen-air flames are thermo-diffusively unstable and burn in localized regions of intensified reaction. The flames propagate by the "active species-diffusion" mechanism, rather than the "thermal propagation mode" adopted by lean methane mixtures. We are interested in understanding mechanisms potentially explaining the influence of hydrogen on the flame structure of hydrogen-methane-air flames. The combustion behavior of hydrogen-methane fuel mixtures is then described with a set of relations derived from the heat release structures. The heat release characteristics of freely propagating flames resemble those of flat steady flames across a broad range of inlet fuel mixtures.

8:39AM MM.00004 Distributed flames and Damköhler "small-scale turbulence" in type Ia supernovae , ANDREW ASPDEN, JOHN BELL, Lawrence Berkeley National Laboratory, STAN WOOSLEY, UC Santa Cruz — High-resolution three-dimensional simulations of carbon-burning flames in type Ia supernovae are used to examine the distributed burning regime (high Karlovitz numbers), or the "small-scale turbulence" regime as it was referred to by Damköhler (1940). He predicted that the turbulent flame speed and width are determined by the nuclear burning time scale and a diffusion coefficient prescribed by the turbulence. The scaling predicts its own breakdown when the turbulent Damköhler number reaches unity. We demonstrate that the simulations are in agreement with the scaling predictions, and propose a method for predicting the limiting flame speed and width based on small-scale simulations. This flame speed can then be used to construct a turbulent flame model to study very large scale distributed supernova flames.

8:52AM MM.00005 Linear stability and long-time evolution of planar lean premixed H2/air flames[1] , C. ALTANTZIS, C.E. FROUZAKIS, S. KERKEMEIER, Aerothermochemistry and Combustion Systems Laboratory, ETH Zurich, Switzerland, A.G. TOMBOULIDES, Dept. of Engineering and Management of Energy Resources, U. W. Macedonia, Kozani, Greece, K. BOULOUCHOS, Aerothermochemistry and Combustion Systems Laboratory, ETH Zurich, Switzerland — In the absence of buoyancy and acoustic interactions, two mechanisms can render premixed propagating flames unstable: hydrodynamic instabilities, stemming from the density jump across the flame, and thermal-diffusive instabilities in subunity Lewis number mixtures. In the present study, the linear stability of lean H_2/air flames as well as their long-term evolution is studied using both single-step and detailed chemistry in 2D rectangular domains of height h=3 to 80 laminar flame thicknesses with periodic boundary conditions imposed along the horizontal boundaries. At the inflow boundary, the lean (equivalence ratio $\phi = 0.6$) mixture enters with a velocity equal to the laminar flame speed, temperature T=298 K and pressure p=5 atm. At these conditions, the Lewis number of the fresh mixture is sufficiently lower than unity (Le=0.4) and the expansion ratio ($\sigma = 6.2$) is large enough so that both mechanisms play a destabilizing role.

[1]Support: Swiss National Science Foundation grant 200021-116669/1.

9:05AM MM.00006 Surface Tension of Premixed Flames as Gasdynamic Discontinuities: Principle and Quantitative Evaluation[1] , HANNES DIRKS, ANDREAS G. CLASS, CHRISTIAN BRUZZESE, Karlsruhe Institute of Technology, Germany — Viewed on hydrodynamic length scales, a typical premixed laminar flame is thin and can be described as a gasdynamic discontinuity separating the fresh mixture from the burned products. We perform an integral momentum analysis of cylindrical premixed flames viewed as gasdynamic discontinuities and quantitatively investigate the jump of momentum at the surface of discontinuity. We interpret the jump of momentum as the result of a localised force and compute the surface tension of the surface of discontinuity, exploiting numerical data and detailed chemical kinetics. Surface tension of premixed flames is shown to be negative. Therefore, in order to establish a stable flame more stringent conditions must be satisfied than one would expect from a pure consideration of the flame speed relation. For numerical simulations, we suggest an implementation based on distributions that implicitly satisfies the momentum balance.

[1]This work was supported by DFG grant SFB 606.

9:18AM MM.00007 Dissolution of a solid particle in a linear velocity field in the presence of chemical reactions[1] , DONNA BLACKMOND, DEVIN CONROY, OMAR MATAR, Imperial College London — The growth or dissolution rate of a particle in a steady linear velocity field is investigated. The particle contains a pure composition of species D and the outer liquid contains species D and L which undergo a reversible reaction. The governing equations are solved asymptotically in the limit of a small Reynolds number based on a particle length scale, with the Prandtle number of order one and a large Schmidt number. We explore the rate of growth or dissolution as a function of the particle curvature, Stefan number, Reynolds number and a scale for the concentration difference.

[1]EPSRC Grant number EP/E027393

9:31AM MM.00008 Visualization of transverse annular jets , PHILIPP BOETTCHER, California Institute of Technology, IOANNIS G. MIKELLIDES, DAVID A. VAUGHAN, NASA-JPL, JOSEPH E. SHEPHARD, JASON DAMAZO, California Institute of Technology — 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 axisymmetric radial jet of variable width impinging 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 reservoir. The flow is then observed using a schlieren system which shows the shock wave from the start-up and the subsequent mixing between the jets. The main parameters under investigation were the reservoir pressure and the area ratio between the axial and radial jet. The effect of these parameters on the qualitative flow behavior is discussed. This effort was carried out in conjunction with modeling efforts at JPL.

9:44AM MM.00009 Estimating soot emissions from an elevated flare[1] , VICTOR ALMANZA, GUSTAVO SOSA, Mexican Institute of Petroleum — Combustion aerosols are one of the major concerns in flaring operations, due to both health and environmental hazards. Preliminary results are presented for a 2D transient simulation of soot formation in a reacting jet with exit velocity of 130 m/s under a 5 m/s crossflow released from a 50 m high elevated flare and a 50 cm nozzle. Combustion dynamics was simulated with OpenFOAM. Gas-phase non-premixed combustion was modeled with the Chalmers PaSR approach and a $\kappa - \varepsilon$ turbulence model. For soot formation, Moss model was used and the ISAT algorithm for solving the chemistry. Sulfur chemistry was considered to account for the sourness of the fuel. Gas composition is 10 % H2S and 90 % C2H4. A simplified Glassman reaction mechanism was used for this purpose. Results show that soot levels are sensitive to the sulfur present in the fuel, since it was observed a slight decrease in the soot volume fraction. NSC is the current oxidation model for soot formation. Predicted temperature is high (about 2390 K), perhaps due to soot-radiation interaction is not considered yet, but a radiation model implementation is on progress, as well as an oxidation mechanism that accounts for OH radical. Flame length is about 50 m.

[1]Tommaso Lucchini, Politecnico di Milano, for providing his Moss and ISAT codes.

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MN Experimental Techniques IV: PIV 200C

8:00AM MN.00001 Strategies for PIV Outlier Replacement using Gappy POD , SAM RABEN, JOHN CHARONKO, Virginia Tech, PAVLOS VLACHOS, Virgnia Tech — This work presents methodologies for reconstructing erroneous measurements in gappy DPIV data using Proper Orthogonal Decomposition (POD). Current methods for data reconstruction using POD require a priori knowledge of the true solution [Venturi and Karniadakis, J. Fluid Mech. 2004]. This limitation renders the method ineffective for reconstructing experimental data. Here, strategies for optimizing Gappy POD reconstruction using different criteria for modal convergence as well as an iteratively reducing point selection algorithm are shown. Gappy flow fields were created using wall turbulence DNS data. Gappyness levels of 5%, 10%, 20%, 50% and, 80% were created with gap sizes of 1x1, 3x3, 5x5, and arbitrary NxM vector spacing. Noise, equivalent to that of DPIV error, was also added. Data reconstruction accuracy was compared against other currently used methodologies, including bootstrapping, kriging, and the universal outlier detection. The gappy POD method presented here is shown to accurately predict the optimum reconstruction with errors on the order of the error associated with basic DPIV velocity measurements.

8:13AM MN.00002 Quantification of particle concentration in sheet illumination imaging techniques , PHILIP KNOWLES, NAVAIR Pax River, KEN KIGER, Univ. of Maryland — In numerous quantitative imaging methods such as PIV, light sheet illumination is used to limit the region of scattered light close to the focal plane of the camera, effectively improving the contrast of the in-focus image. It is often desirable to use these imaging methods not only to provide displacement information, but also additional metrics such as particle concentration as well. At its simplest level, the quantification of a dispersed particle concentration would seem straightforward: simply count the particles in the image and divide by the interrogation area and light sheet thickness. In practice, however, this typically only yields order-of-magnitude estimates of the concentration. Particle identification within the image will inherently depend on the image size and intensity, which in turn depends on the imaging optics, local illumination and particle position/geometry. In the current work, we demonstrate the influence of the light sheet profile (focusing/attenuation), scattering by tracer particles and wall reflections on determining the effective measurement volume. An empirical calibration method is presented to account for these effects, which allows for quantification of the concentration to within 15%. Application of this method to a multiphase suspension is demonstrated. This work is supported by the NSF under grant 0351443 and AFOSR grant FA95500810406.

8:26AM MN.00003 ABSTRACT WITHDRAWN –

8:39AM MN.00004 Cross-correlation PIV using infrared and visible Nd:YAG pulses and a Nikon dSLR , MICHAEL HALLBERG, PAUL STRYKOWSKI, University of Minnesota — A cross-correlation particle image velocimeter is assembled using two Nd:YAG lasers and a Nikon D70 SLR camera. One laser is frequency doubled while the other is not thereby producing a pulse train of two separate colors, green and IR respectively. The internal IR filter is removed from the Nikon D70 and the resulting CCD is sensitive over a range through the visible and into the IR (up to at least 1064nm). Each image was then exposed to both laser pulses revealing two distinct particle images, one green and the other red. The doubly exposed images were separated into two images corresponding to the green channel and the red channel thereby allowing cross-correlation. Initially, the correlation was contaminated by the color filter used to produce the RGB colors on a single CCD, generating a large zero displacement peak. Image processing was employed to reduce the zero peak; the next highest peak generally yielded the particle displacements. Camera manufacturers are developing technologies that will supplant the current color filter (cf. Sigma's image sensor), hopefully making the dSLR a viable option for PIV going forward.

8:52AM MN.00005 In vivo μPIV measurements of blood velocity in small vessels of a rat model , CHIA MIN LEONG, RPI, JOHN RUSSELL, NADINE CONNOR, Univ. Wisconsin Medical School, MARKUS HONKANEN, TUT, TIMOTHY WEI, RPI — Aging-related muscular changes have been shown to affect voice production. There is correlation between muscular changes and changes in capillary hemodynamics and structure with aging. Alterations in oxygen transport to cells and tissues at the capillary level has been hypothesized as one of the key factors that causes muscular changes thus voice production. Since oxygen transport is related to hemodynamics, we start by measuring blood velocity in capillaries of cremaster muscle of a living rat. The μPIV technique is adapted for measuring blood velocity where red blood cells are used as 'seeding particles'. The accuracy of the μPIV measurements are determined by comparison with results obtained using other techniques such as particle tracking velocimetry (PTV). Finally, challenges in measuring flow through three-dimensional larynx geometry will be discussed.

9:05AM MN.00006 See Through the Static: 3D Synthetic Aperture PIV , JESSE BELDEN, MIT, TADD T. TRUSCOTT, NUWC, ALEXANDRA H. TECHET, MIT — A new method for resolving three-dimensional (3D) fluid velocity fields using a technique called synthetic aperture particle image velocimetry (PIV) is presented. The method makes use of the lightfield imaging and synthetic aperture refocusing techniques that are emerging in the imaging community. Images are captured using an array of cameras positioned on one plane such that the fields of view of the cameras overlap and images can be easily recombined in software using a warp-shift-average algorithm to digitally refocus on different planes. The result is sharply focused particles in the plane of interest, whereas particles out-of-plane appear blurred. The 3D intensity field of particle-laden flows can be reconstructed by refocusing throughout the entire volume and filtering out the blurred particles. 3DPIV techniques can then be applied to these intensity fields to extract velocity data. This technique shows the potential of enabling larger volumes to be resolved with more particles, yielding higher spatial resolution than existing methods. A simulated vortex ring flow field demonstrates the capability of the technique for resolving vector fields in 3D.

9:18AM MN.00007 ABSTRACT WITHDRAWN –

9:31AM MN.00008 3-D Velocity Measurements Around an Optically Suspended Sphere , CHRIS HINOJOSA, JEREMIAH ZIMMERMAN, NATHALIE NEVE, DEREK TRETHEWAY, Portland State University — The 2-D velocity fields at the mid-plane of an optically trapped sphere can be obtained using the μPIVOT (Neve et al. 2008), an integrated optical tweezers (OT) and micron-resolution particle image velocimetry (μPIV) instrument. Typically, the optical trap location is centered in the μPIV measurement plane and their movements coupled. However, positioning lenses in the path of the OT laser allow the optical trap to be moved independently of the measurement plane. In order to suspend a sphere at a fixed point in space while measuring velocities around the sphere, the positioning lenses must compensate for the movement of the measurement plane. In this work, the relationship between positioning lens movement and trap location is determined by measuring the settling time of a sphere initially positioned out of focus. With this relationship determined, 2-D velocity fields are measured at different planes around a sphere in uniform flow. The measured velocity fields are compared to analytical and computational predictions to examine the effects of optical tweezers on μPIV tracer particle motion and to validate the 3-D velocity measurement potential of the μPIVOT.

9:44AM MN.00009 Nano-Velocimetry for Nanofluidics[1] , CUIFANG KUANG, GUIREN WANG, NANOFLUIDICS AND MICROFLUIDICS LAB TEAM — In order to understand physical and biological phenomena in nanoscale and build functional and practical nanofluidic devices, it is important to know the flow velocity profile. Due to the Abbe's diffraction limit barrier, traditional optical methods have so far failed in measuring the velocity profile in a nanochannel. Atomic force microscopy cannot be used for nanochannels without an opened sidewall. We have, for the first time, been able to measure the flow velocity profile for nanofluidics with a spatial resolution better than 70 nm. A novel optical point measurement method is presented, which applies Stimulated Emission Depletion (STED) to Laser Induced Fluorescence Photobleaching Anemometer (LIFPA) techniques to measure flow velocity. Herein we demonstrate this far-field nanoscopic velocimetry by measuring the velocity profile in a nanocapillary with an inner diameter of 360 nm. The closest measuring point from the wall is about 35 nm. The velocity can clearly be differentiated within just a 20 nm step, even near the axial region of the nanocapillary. This method opens up a new class of functional measuring techniques for nanofluidics and for near wall flows.

[1]This work has been financially supported by the NSF RII funding (EPS-0447660).

9:57AM MN.00010 A Time-Resolved PIV with continuous Laser and High Frame Rate Camera , AMIR ELZAWAWY, YIANNIS ANDREOPOULOS, City College of New York — Monitoring the evolution of turbulent structure using PIV requires time resolved measurements. Existing time-resolved PIV systems are limited mainly by the pulsed laser repetition rate, which currently is at about 10 KHz. In the present work we explore the possibility of using a continuous laser and a camera with a frame rate in the order of MHz, with limited number of frames. This set-up has been applied to an incompressible turbulent boundary layer flow configured in a low speed wind tunnel. In order to evaluate the technique and particularly investigate the effect of exposure time, several experiments were performed and the results compared with pulsed laser PIV and hot-wire results. Exposure times up to 15 per cent of the time between frames were used. The effect of the exposure time on the turbulence intensity was also investigated.

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MP Instability: Rayleigh-Taylor 200D

8:00AM MP.00001 Experimental Study of Rayleigh-Taylor Instability Using Paramagnetic Fluids , VLADIMER TSIKLASHVILI, OLEG LIKHACHEV, JEFFRY JACOBS, University of Arizona — Experiments that take advantage of the properties of paramagnetic liquids are used to study Rayleigh-Taylor instability. A gravitationally unstable combination of a paramagnetic salt solution and a nonmagnetic solution is initially stabilized by a magnetic field gradient that is produced by the contoured pole-caps of a large electromagnet. Rayleigh-Taylor instability originates with the rapid removal of current from the electromagnet, which results in the heavy liquid falling into the light liquid due to gravity and, thus, mixing with it. The mixing zone is visualized by back-lit photography and is recorded with a digital video camera. For visualization purposes, a blue-green dye is added to the magnetic fluid. The mixing rate of the two liquids is determined from an averaged dye concentration across the mixing layer by means of the Beer-Lambert law. After removal of the suspending magnetic field, the initially flat interface between the two liquids develops a random surface pattern with the dominant length scale well approximated by the fastest growing wavelength in accordance with the viscous linear stability theory. Several combinations of paramagnetic and nonmagnetic solutions have been considered during the course of the research. A functional dependence of the mixing layer growth constant, α, on the properties of the liquids is a primary subject of the present study.

8:13AM MP.00002 The Experimental Study of Rayleigh-Taylor Instability using a Linear Induction Motor Accelerator , NICHOLAS YAMASHITA, JEFFREY JACOBS, University of Arizona — The experiments to be presented utilize an incompressible system of two stratified miscible liquids of different densities that are accelerated in order to produce the Rayleigh-Taylor instability. Three liquid combinations are used: isopropyl alcohol with water, a calcium nitrate solution or a lithium polytungstate solution, giving Atwood numbers of 0.11, 0.22 and 0.57, respectively. The acceleration required to drive the instability is produced by two high-speed linear induction motors mounted to an 8 m tall drop tower. The motors are mounted in parallel and have an effective acceleration length of 1.7 m and are each capable of producing 15 kN of thrust. The liquid system is contained within a square acrylic tank with inside dimensions 76 x76x184 mm. The tank is mounted to an aluminum plate, which is driven by the motors to create constant accelerations in the range of 1-20 g's, though the potential exists for higher accelerations. Also attached to the plate are a high-speed camera and an LED backlight to provide continuous video of the instability. In addition, an accelerometer is used to provide acceleration measurements during each experiment. Experimental image sequences will be presented which show the development of a random three-dimensional instability from an unforced initial perturbation. Measurements of the mixing zone width will be compared with traditional growth models.

8:26AM MP.00003 Experimental study of the Rayleigh-Taylor instability at multimode interfaces , JEREMY WHITE, JASON OAKLEY, MARK ANDERSON, RICCARDO BONAZZA, University of Wisconsin-Madison — The gravitationally driven 2-D Rayleigh-Taylor (RT) instability is studied experimentally using simple, quantifiable multimode interfaces for two different Atwood numbers, A=0.46 and A$\sim$ 1. This study is performed using a magnetic fluid suspension technique that allows for precise interfacial shaping by exploiting the unique properties of magnetorheological (MR) fluids. The multimode shapes examined include a handful of modes which were chosen to minimize the effects of the physical test section size and surface tension on the development of the instability. A high speed X-ray radiography based diagnostic system is used to measure the evolution of the RT bubbles and spikes. The method for prescribing the initial condition allows for individual modes, which are chosen a priori, to be tracked for studying saturation, merger, and their collective influence on the overall mixing width.

8:39AM MP.00004 Two-Wheel Experiment for detailed measurements of Rayleigh-Taylor Turbulence[1] , AARON HALEY, RAGHU MUTNURI, ARINDAM BANERJEE, Missouri S&T — A novel two-wheel experiment to investigate incompressible turbulent Rayleigh-Taylor (RT) instability is described. The experiment consists of two counter rotating wheels, placed side by side, such that the axes of the wheels are normal to gravity. A test section, carrying a pair of immiscible fluids, is loaded on one wheel such that the heavier fluid is held radially outwards ensuring a stable stratification (no mixing). The test section is then oscillated to impose controlled multi-mode initial perturbations on the interface and finally transferred to the adjacent wheel using a solenoid actuated transfer mechanism. Upon transfer, the fluid stratification in the test section is reversed which leads to development of RT instability. Large centrifugal accelerations (10g) produced by the rotation of the wheels allow investigation of late time RT turbulence. Details of the mixing layer development and growth constants are captured using high speed backlit imaging. A variety of immiscible fluid combinations are utilized to investigate development of the RT mixing over a large range of Atwood numbers (0.1-0.99) and the results are compared with similar data available in the literature.

[1]Authors acknowledge financial support from LANL subcontract 73697-001-09 and University of Missouri Research Board.

8:52AM MP.00005 Effect of inital conditions in low-Atwood Rayleigh-Taylor Mixing , YUVAL DORON, ANDREW DUGGLEBY, Texas A&M University — The effects of interface initial conditions in low-Atwood number Rayleigh-Taylor mixing are reported. The low-Atwood number water channel at Texas A&M is modified with a servo motor controlled flapper device at the end of a splitter plate. The familiar bubble and spike phenomena for different wave numbers are observed. Average mixing rates are measured optically with the application of Beer-Lambert law and are used to measure the effect of initial conditions on the mixing height growth rates. Results show that single mode initial conditions all achieve the same growth rates within uncertainty. Details of the experimental set up are included as well as a discussion of future work.

9:05AM MP.00006 Hybrid WENO/Central Difference Navier-Stokes Simulation of Rayleigh-Taylor Instability[1] , WAI-SUN DON[2], Brown University, OLEG SCHILLING, Lawrence Livermore National Laboratory — A new hybrid weighted essentially non-oscillatory (WENO)/central finite-difference method has been developed for the high-resolution, multi-dimensional, efficient simulation of turbulent mixing induced by interfacial hydrodynamic instabilities. Multi-resolution analysis is used to dynamically determine regions in which large gradients or discontinuities exist (where upwinding is applied) and regions in which the flow is relatively smooth (where central differencing is applied). This method is used to solve the fluid dynamics equations describing Rayleigh–Taylor unstable flow at intermediate and large Atwood number, and is shown to be robust for large initial density contrasts. Comparisons of the mixing layer widths, molecular mixing parameter, energy spectra, and other quantities are used to explore the effects of Atwood number on the evolution of turbulence statistics.

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

[2]also at Hong Kong Baptist University

9:18AM MP.00007 Wavelet-Based Simulations of Single-Mode Rayleigh-Taylor Instability[1] , SCOTT J. RECKINGER, DANIEL LIVESCU, OLEG V. VASILYEV — The single-mode compressible Rayleigh-Taylor instability is investigated using numerical simulations on an adaptive mesh, performed with the Adaptive Wavelet Collocation Method (AWCM). Due to the physics-based adaptivity and direct error control of the method, AWCM is ideal for resolving the wide range of scales present in the development of the instability. The problem is initialized consistent to the solutions to the linear stability theory. Of interest are the departure time from the linear growth, the onset of strong non-linear interactions, and the late-time behavior of the fluid structures. The late time buble/spike velocities are computed and compared to those obtained in the incompressible case.

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

9:31AM MP.00008 A Three- or Four-Equation Reynolds-Averaged Navier-Stokes Model of Large Reynolds Number Rayleigh-Taylor Turbulence and Mixing[1] , OLEG SCHILLING, Lawrence Livermore National Laboratory, GREGORY BURTON, Stanford University — Using data from a 3072^3 direct numerical simulation of Rayleigh–Taylor flow [*Nature Physics* **2**, 562 (2006)], it is shown *a priori* that gradient-diffusion and scale-similarity closures provide a closed three- or four- equation Reynolds-averaged Navier–Stokes model that correlates well with the data. In particular, using order of magnitude estimates of the exact transport equations and their closures, it is shown that the turbulent production and destruction terms in the turbulent kinetic energy dissipation rate and density variance dissipation rate equations scale as the square root of the turbulent Reynolds number, resulting in scale-similarity model coefficients that asymptote. A simplified algebraic Reynolds stress tensor model, similar to that used in turbulent convection and other buoyancy-driven turbulent flows, is shown to provide a good model for the anisotropic Reynolds stress tensor. Exploration of other algebraic Reynolds stress modeling approaches for incorporating the early-time nonequilibrium production-to-dissipation mechanisms is also discussed.

[1]This 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 MP.00009 The "second-wind" phenomenon in single-wavelength Rayleigh-Taylor , PRAVEEN RAMAPRABHU, KARTHIK MUTHURAMAN, UNC Charlotte, GUY DIMONTE, Los Alamos National Laboratory, PAUL WOODWARD, University of Minnesota, CHRIS FRYER, Los Alamos National Laboratory, YUAN-NAN YOUNG, NJIT, SUNG-IK SOHN, Kangnung National University — The late-time, single-mode Rayleigh-Taylor (RT) flow asymptotes to a Froude number approaching 1, higher than predicted by potential flow models. The reacceleration to a higher terminal velocity [1] appears to be triggered by the appearance of Kelvin-Helmholtz (KH) vortices at small Atwood numbers. At large density differences, the KH instability is stabilized, with the result that the terminal velocity is in agreement with Layzer-type models. We compare results from simulations using multiple codes, with recently published experiments of [2], and with a simple model. The appearance of KH is also complicated by the presence of additional effects such as viscosity and surface tension. The results are of relevance to bubble- competition models of fully turbulent RT. [1] Ramaprabhu, P., et al. 2006, Physical Review E. 74, 066308. [2] Wilkinson, J.P. & Jacobs, J.W. 2007, Phys. Fluids 19, 124102.

9:57AM MP.00010 Rayleigh-Taylor Instability in Nonlinear Schrödinger Flow , SHU JIA, JASON W. FLEISCHER, Princeton University — We consider the Rayleigh-Taylor instability in nonlinear Schrödinger flow. In this superfluid-like case, wave diffraction, rather than viscosity or surface tension, sets the spatial scale for long-wave growth. Theoretically, we apply a polar (Madelung) transformation to the complex wavefunction and map intensity to density and velocity to the gradient of the phase. We show analytically that, unlike the instability dynamics in normal fluids, the superfluid behavior is strongly nonlinear and compressible from the start. Experimentally, we demonstrate the instability all-optically in a photorefractive crystal, using a self-defocusing nonlinearity as an effective pressure and a refractive index gradient as the driving acceleration. Observations of the characteristic spatial period show excellent agreement with scaling calculations from perturbation theory. We find that density fingering is always accompanied by vortex generation and that pressure effects strongly influence the finger period and mixing depth. The results hold for any Schrödinger fluid, e.g. superfluids and quantum plasma, and lay the foundation for a variety of fluid-inspired instabilities in nonlinear optics.

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MQ Instability: General 200E

8:00AM MQ.00001 Nonlinear transient growth and detecting the laminar-turbulent boundary , RICH KERSWELL, CHRIS PRINGLE, Bristol University — The area of transition to turbulence has experienced a spurt of activity recently in wall-bounded shear flows where transition is triggered by finite amplitude disturbances. This has been stimulated by the discovery of new solutions to the governing equations which are disconnected from the steady unidirectional "base" flow in, for example, pipe flow and plane Couette flow. Understanding how all this new (nonlinear) work fits together with transient growth ideas which focus on linear mechanisms based around the base state presents an interesting challenge. In this talk, we attempt to bridge the gap from linear optimals to the (very nonlinear) laminar-turbulent boundary in phase space which determines whether a given disturbance will trigger turbulence or not.

8:13AM MQ.00002 Transient growth in Rayleigh-Benard-Poiseuille and Rayleigh-Benard-Couette flows , JOHN SOUNDAR JEROME JOSEPH, JEAN-MARC CHOMAZ, PATRICK HUERRE — A study on the effect of buoyancy in plane Poiseuille flow and Couette flow has been carried out through linear temporal stability analysis, in the normal and non-normal framework. Here, the buoyancy is solely induced by a constant thermal stratification in the wall-normal direction. In the context of linear temporal stability analysis, a parametric study of the marginally stable modes with respect to Reynolds and Rayleigh numbers has been presented. It is found that the wall-normal temperature gradient has a negligible effect on the Tollmien-Schlichting instability. Nevertheless, the nonmodal analysis shows that the coupling between the body force and shear has a favorable effect on the transient growth of longitudinal disturbances in both plane Poiseuille flow and Couette flow. The transient growth characteristics have been illustrated by making use of different energy norms.

8:26AM MQ.00003 Flow induced oscillation of a cylinder in a Hele-Shaw cell , B. SEMIN, H. AURADOU, J.-P. HULIN, FAST Laboratory, CNRS, Pierre et Marie-Curie Paris 6 and Paris-Sud 11 Universities (France), A. DECOENE, V. SCELLES, Orsay Maths Department, CNRS, Paris-Sud 11 University (France), A. LEFEBVRE, CMAP, CNRS, Ecole Polytechnique (France) — Spontaneous regular oscillations of a confined cylinder in a steady Poiseuille flow are observed down to small Reynolds numbers ($Re = 15$). In this study, the cylinder is perpendicular to the mean flow, parallel to the walls of a Hele-Shaw cell and free to move only in the direction perpendicular to them; the ratio of the diameter of the cylinder by the cell aperture is 0.7. Experimentally, the cylinder is held by long thin threads. This flow-structure coupling, resulting from the confinement, has also been modelled successfully using $2D$ finite elements simulations. The oscillations are quasi-sinusoidal in a wide range of Re value (Re is defined using the mean velocity and the diameter of the cylinder). The threshold value ($Re = 15$) is much smaller than for classical vortex shedding past a nonconfined cylinder ($Re = 45$). The amplitude increases with the Reynolds number until saturation. The frequency increases almost linearly with Re (Strouhal number close to 1) even when contact with the walls occurs; it increases with the diameter of the cylinder and decreases with its density.

8:39AM MQ.00004 Transient growth in Core-Annular Flow , GENNARO COPPOLA, DETEC, University of Naples Federico II, ANNAGRAZIA ORAZZO, LUIGI DE LUCA, DIAS, University of Naples Federico II — The classical problem of the stability of Core-Annular Flow (CAF) in pipes is here reconsidered from the point of view of linear non modal analysis. An accurate Chebyshev pseudospectral code in polar coordinates has been developed in order to separately discretize the two phases of the flow. The classical tools of non modal analysis have been employed in order to assess the effects of non normality on transient evolution of small perturbations to core-annular base flow. Linear transient growths of three dimensional perturbations are computed for typical potentially stable configurations, by taking into account the effects of viscosity and volume ratios between the two liquids, as well as of Reynolds number and surface tension. A detailed numerical investigation has been conducted on wide regions of the parameters space and the occurrence of substantial transient growth has been found for asymptotical stable configurations. Optimal perturbations for the most critical parameters have been determined and their structure during the unsteady evolution has been considered. It is found that in situations in which axisymmetric modes of disturbance are expected to constitute the most dangerous ones, in subcritical conditions spiral disturbances can provide higher levels of energy amplification.

8:52AM MQ.00005 The Effect of Brownian Motion of Nano-Catalysts on the Stability of Reactive Fronts in Porous Media , KARIM GHESMAT, HASSAN HASSANZADEH, JALAL ABEDI, JOHN CHEN, Department of Chemical Engineering, University of Calgary — The reactive flows in porous media that involve the displacement of fluids by different physical properties may lead to a hydrodynamic instability. The use of nano-particles as catalysts in porous media has recently been increased and is generally relevant to applications that include *in-situ* heavy oil upgrading and removal of reactive and non-reactive pollutants in groundwater. The objective is to investigate the effects of *nano-catalysts* and *chemical reactions* on this instability. In order to understand the physics of this flow displacement, the basic equations of conservation of mass and momentum are linearized and solved numerically for a homogenous porous medium. The analysis reveals that increasing the reaction rate enhances the instability around an interface including nano-catalysts while increasing the nano-catalysts deposition rate in porous media usually stabilizes the front. The effects of the interface sharpness, nano-particle diffusion coefficient, permeability of porous media, and viscosity ratios of different phases will also be discussed.

9:05AM MQ.00006 Experiments on two- and three-dimensional vortex flows in lid-driven cavities , TANJA SIEGMANN-HEGERFELD, Institute of Fluid Mechanics and Heat Transfer, Vienna University of Technology, STEFAN ALBENSOEDER, CeBeNetwork GmbH, HENDRIK C. KUHLMANN, Institute of Fluid Mechanics and Heat Transfer, Vienna University of Technology — Vortex flows in one-sided lid-driven cavities with different cross-sectional aspect ratios ($\Gamma = 0.26$ up to $\Gamma = 6.3$) are investigated experimentally. In all cases the spanwise aspect ratio $\Lambda >> \Gamma$ is very large and much larger than most previous experiments. Flow-structure visualizations will be presented together with quantitative LDA and PIV measurements. The experimental results are in good agreement with the critical data from numerical stability analyses and with nonlinear simulations. Experimentally, we find four different three-dimensional instabilities. Particular attention is paid to the so-called C_4 mode which arises at large cross-sectional aspect ratios. When the spanwise aspect ratio is small the first bifurcation of the C_4 mode is strongly imperfect.

9:18AM MQ.00007 Thickness dependence of drag and kinematic viscosity in quasi-two-dimensional flows[1] , PAUL W. FONTANA, EDWARD TITMUS, ADRIAN KIRN, Seattle University — In many experimental and natural quasi-two-dimensional (Q-2D) flows the effects of internal viscosity and surface friction are significant but difficult to distinguish. We have made precise, independent measurements of both kinematic viscosity and coefficient of external drag as functions of film thickness in a Q-2D experiment using soap films in a circular Couette cell configuration, using a combination of vortex decay rates and steady-state shear lengths. Both the kinematic viscosity and the drag coefficient are found to depend inversely on film thickness, as expected theoretically. This result is enabling quantitative experimental study of stability thresholds in Q-2D basic flows.

[1]Supported by Seattle University and the M. J. Murdock Charitable Trust.

9:31AM MQ.00008 The structure of the transition boundary for shear flows , NORMAN LEBOVITZ, University of Chicago — The shape and properties of the basin of attraction of the stable laminar point is investigated for finite-dimensional models of shear flows. In some of these models, the basin boundary is the stable manifold of an equilibrium point XIb, the lower-branch point. As parameters change, the boundary undergoes a topological change at which a periodic orbit P emerges via a homoclinic bifurcation, and thereafter the major part of the basin boundary coincides with the stable manifold of P. The stable manifold of XIb is then detectable only as an "edge," i.e., the boundary between sets having different relaminarization properties. Implications for the nature of the edge are discussed.

9:44AM MQ.00009 The Formation of Packets of Hairpins in Shear Flows[1] , JACOB COHEN, MICHAEL KARP, Technion, Faculty of Aerospace Engineering, ILIA SHUKHMAN, Russian Academy of Sciences, Irkutsk — In the present work we utilize a recently developed new method in an attempt to understand the generation of packets of hairpin vortices from a pair of counter rotating streamwise vortices embedded in uniform shear flow. This analytical-based solution method is capable of following (numerically) the evolution of finite-amplitude localized vortical disturbances embedded in shear flows. Due to their localization in space, the surrounding base flow is assumed to have homogeneous shear to leading order. The method can solve in a novel way the interaction between a general family of unbounded planar homogeneous shear flows and any localized disturbance. The solution is carried out using Lagrangian variables in Fourier space which is convenient and enables fast computations. The revealed mechanism for generation of packets of hairpins seems to be universal and has been observed in the past both in fully developed wall-bounded shear flows as well as in wall-bounded transitional shear flows.

[1]This research was supported by the Israeli Science Foundation under Grant No. 1247/06.

9:57AM MQ.00010 Lock exchange and autocatalytic reaction front , IBRAHIM BOU MALHAM, CNRS, NOLWENN JARRIGE, UPMC Paris6, JEROME MARTIN, CNRS, NICOLE RAKOTOMALALA, UPMC Paris6, LAURENT TALON, CNRS, DOMINIQUE SALIN, UPMC Paris6 and IUF — Lab FAST, UPMC Paris6 UPS Paris11 and CNRS, Bat. 502, Orsay, F-91405, France. Viscous lock-exchange is the reciprocal exchange of two fluids of different density in a horizontal channel: the front between the two fluids spreads as the square root of time with a diffusion coefficient reflecting the buoyancy, viscosity and geometry of the current. Iodate Arsenous Acid (IAA) autocatalytic reaction front between reactant and product propagates as solitary waves (constant velocity and stationary concentration profile) resulting from a balance between molecular diffusion and chemical reaction. The fluid left behind the front has a different density leading to a lock-exchange configuration. We revisit, with chemical reaction, the lock exchange situation. We present an experimental analysis of buoyancy effects on IAA reaction fronts, propagating in horizontal rectangular channels of different aspect ratio. We observe stationary-shaped fronts, spanning the height of the cell and propagating along the cell axis. As the result of the data analysis, we find out that the product of the front velocity times its extension is nothing but the diffusion coefficient of the lock-exchange in the absence of chemical reaction. The analysis accounts for results in other geometries and for other chemical reactions.

Tuesday, November 24, 2009 8:00AM - 9:31AM –

Session MR Rotating Flows 200F

8:00AM MR.00001 Dynamics of a fluid inside a precessing cylinder , ROMAIN LAGRANGE, PATRICE MEUNIER, CHRISTOPHE ELOY, IRPHE, FRANÇOIS NADAL, CEA/CESTA — The instability of a fluid inside a precessing cylinder is studied theoretically and experimentally, motivated by aeronautical and geophysical applications. Precessional motion forces hydrodynamic waves called Kelvin modes whose structure and amplitude are predicted by a linear inviscid theory. When a forced Kelvin mode is resonant, its amplitude diverges (and saturates due to viscous effects), which makes the flow unstable for sufficiently high Reynolds numbers. 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.

8:13AM MR.00002 Rotating spin-up in three dimensions , JIN-QIANG ZHONG, Yale University, MICHAEL PATTERSON, Yale University & University of Bath, UK, JOHN WETTLAUFER, Yale University — Motivated by previous studies we revisit the well known transient spin up problem, with and without buoyancy forcing, using a new experimental technique. In so doing, we are able to examine the fluids behavior by extracting a series of two-dimensional velocity fields at multiple levels simultaneously thereby producing a three-dimensional view of the evolving flow. The results provide startling detail of the rich three dimensional structure of the flow, and reveal, amongst other things, depth dependent axisymmetric pattern formation and breakdown. Vertical slices of the velocity field reveal highly localized up and downwellings. Our quantitative probing produces continuous velocity fields amenable to direct comparison with simulations.

8:26AM MR.00003 Large scale flow instabilities in rotating flows , L. DEL CASTELLO[1], H.J.H. CLERCX, R.R. TRIELING, A. TSINOBER[2] — Background rotation affects the dynamics of fluid flows with a combination of linear and nonlinear effects, according to the relative importance of the Coriolis acceleration compared to the convective and forcing terms in the Navier-Stokes equation. We perform experiments on a turbulent flow ($Re_\lambda \sim 150$) electromagnetically forced in a confined tank put on a rotating table, and we measure the flow using Particle Tracking Velocimetry. We focus here on the anisotropic effects of rotation at the large scales, and we define the Rossby number $Ro_f = \frac{U_f^2/L}{2\Omega U_f}$ based on the forced velocity scale U_f. With a mild rotation of $0.2\ \mathrm{s}^{-1}$ ($Ro_f \sim 1.8$), we observe an enhancement of horizontal and vertical velocity gradients of the large scale flow, but the overall forced flow pattern remains stable. When the rotation is increased to $2.0\ \mathrm{s}^{-1}$ ($Ro_f \sim 0.2$), we observe instead the dampening of the velocity gradients at large scales, while the forced flow pattern strongly fluctuates. With a strong rotation of $5.0\ \mathrm{s}^{-1}$ ($Ro_f \sim 0.07$), the stability of the forced mean flow structures is finally restored.

[1]Dept. of Physics, Eindhoven University of Technology, P.O.Box 513, 5600MB Eindhoven, The Netherlands
[2]Inst. Math. Sciences and Dept. of Aeronautics, Imperial College London, United Kingdom

8:39AM MR.00004 Effect of the Sign of Ro on the Stability of Stewartson Layers Generated by a Rotating Disc , JOZEF H.A. VLASKAMP, PETER J. THOMAS, University of Warwick, Fluid Dynamics Research Centre, RAINER HOLLERBACH, University of Leeds, Department of Applied Mathematics — The effect of the sign of rotation on the stability of the Stewartson layer formed by a rotating disc in a rotating fluid is investigated experimentally and numerically. A discrepancy exists between two earlier studies. Ref. [1] found the sign of Ro to have no influence, while Ref. [2] found a strong asymmetric behaviour. Numerical investigations in Ref. [3] suggested a difference in the boundaries along the disc as a possible cause of the discrepancy between the studies. In the current study experimental work is performed on the large scale facility at the University of Warwick (overall height 5.7m and 1.4 m diameter), with a single disc inside a movable end-wall. The size of the facility allows a much larger height to radius ratio than in previous experiments. A fully automated, traverse mounted 3D PIV system is used to allow data- acquisition over the length of the Taylor column without the need for recalibration. Further numerical work is conducted using the code described in Ref. [3]. REFERENCES: [1] W. G. Früh,. P.L. Read, J. Fluid Mech., vol. 383, p. 143, 1999. [2] R. Hide, R., C.W. Titman J. Fluid Mech., vol. 29, p. 39, 1967. [3] R. Hollerbach, J. Fluid Mech., vol. 492, p. 289, 2003.

8:52AM MR.00005 Anisotropy properties and geometrical scale-dependent statistics of sheared and rotating turbulence , FRANK JACOBITZ, Mechanical Engineering Program, University of San Diego, San Diego, CA, USA, KAI SCHNEIDER, M2P2-CNRS & CMI, Aix-Marseille University, Marseille, France, WOUTER BOS, LMFA-CNRS, Ecole Centrale de Lyon, University of Lyon, Ecully, France, MARIE FARGE, LMD-IPSL-CNRS, Ecole Normale Superieure, Paris, France — The anisotropy properties of homogeneous turbulence with mean shear and system rotation are studied using both conventional and wavelet-based anisotropy measures. The study is based on a series of nine direct numerical simulations in which the rotation ratio f/S of Coriolis parameter to shear rate is varied. The presence of rotation stabilizes the flow, except for a narrow range of rotation ratios $0 < f/S < 1$. The main mechanism for the destabilization is an increased turbulence production due to increased anisotropy. This anisotropy at large and small scales is quantified by applying conventional measures, such as the Reynolds stress and the dissipation rate anisotropy tensors, respectively. Recently introduced directional wavelet based measures are also applied and compared with the classical ones.

9:05AM MR.00006 Small-scale Statistics of Turbulence induced by Vortices[1] , KELKEN CHANG, GREGORY P. BEWLEY, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization, INTERNATIONAL COLLABORATION FOR TURBULENCE RESEARCH COLLABORATION — We report measurements of the small-scale statistics of a moderate Reynolds number (up to a Taylor microscale Reynolds number of 500) turbulent flow induced by large-scale vortices. We study the flow using Lagrangian particle tracking technique, in which the three-dimensional motion of neutrally bouyant oil particles in air is followed optically using multiple high speed cameras. We compare the results with experimental measurements obtained in a nearly homogeneous and isotropic turbulent flow at comparable Reynolds number.

[1]This work is supported by the Max Planck Society.

9:18AM MR.00007 ABSTRACT WITHDRAWN –

Tuesday, November 24, 2009 8:00AM - 9:57AM –

Session MS Geophysical: General I 200G

8:00AM MS.00001 Laboratory scale simulation of spontaneous vertical convective vortex generation[1] , ALBERT SHARIFULIN, Perm State Technical University, ANATOLY POLUDNITSIN, Perm State University — The new mechanism of spontaneous vertical vortex generation in stratified fluid is under consideration. This phenomenon was discovered in the framework of experimental attempt [1] to proof the hypothesis of universal character of bifurcation curve formulated in [2]. The experiment with slow cubic cell inclination from bottom heating position was performed. The theoretically predicted curve form had been proved; but in the transition process from abnormal convection flow to normal one during bifurcation curve crossing the unexpected spontaneous vertical convective vortex motion has been discovered. Possibility of spontaneous vertical convective vortex generation application to atmospheric behavior explanation and to Earth's mantle one is discussed. New non-local hurricane generation mechanism and observed oceanic volcano archipelago's form explanation attempt are formulated and speculated. [1] AN Sharifulin, AN Poludnitsin, AS Kravchuk Laboratory Scale Simulation of Nonlocal Generation of a Tropical Cyclone. Journal of Experimental and Theoretical Physics, 2008, V.107, No.6, p.1090. [2] AI Nikitin, AN Sharifulin, Concerning the bifurcations of steady-state thermal convection regimes in a closed cavity due to the Whitney folding-type singularity. Heat Transfer – Soviet Research, v.21, no.2, 1989, p.213.

[1]This work was partially supported by RFBR grant 07-01-96070.

8:13AM MS.00002 Frontal instabilities and waves in a differentially rotating two-layer fluid , JAN-BERT FLOR, HELENE SCOLAN, Laboratoire des Ecoulements Geophysiques et Industriels (LEGI) — Fronts are key structures for ocean and atmosphere dynamics and relevant for weather forecasts and climate. In this study, we have investigated the stability of a baroclinic front, generated in a rotating salt-stratified two-layer fluid by a rotating lid at the surface of the fluid. In the parameter space set by rotational Froude number, dissipation (i.e. spin-down to disk rotation time-ratio) and Rossby number, different flow regimes are observed, ranging from axisymmetric, Kelvin Helmholtz instability to irregular baroclinic instable flows, in coherence with former results obtained for two- layer flows in immiscible fluids. New is the evidence of the Rossby-Kelvin instability, which occurs as a consequence of the resonant interaction between Rossby and Kelvin waves. In addition, locally emitted waves with a short wave length are observed in the baroclinic unstable regime. Some of these waves appear for relatively high Richardson numbers, but for wave numbers for which one may also expect Holmboe waves. We discuss these waves in the light of spontaneously emitted inertia-gravity waves and other possible mechanisms.

8:26AM MS.00003 Trapping and hopping of a large "continent" over a thermally convecting fluid , JUN ZHANG, Dept. of Physics and Courant Institute, New York University, JIN-QIANG ZHONG, Dept. of Geology and Geophysics, Yale University, BIN LIU, Courant Institute, New York University — In an annular geometry, we study the interactions between a freely moving, floating boundary and a thermally convective fluid underneath. This experiment aims to investigate the essentials of interplay between large continents and a convective mantle. We find that the continent, depending on its relative size, can be trapped over a convergent or divergent flow at the continent-mantle interface. Such trapped states are regularly interrupted as the continent hops from one trapping position to the other. This rich dynamics can be understood by the heat buildup beneath the continent, a phenomenon referred to as the "thermal blanket" effect in geophysics.

8:39AM MS.00004 Melt Conduit Instability , JOHN WHITEHEAD, Woods Hole Oceanographic Institution, MIRANDA HOLMES-CERFON, Courant Institute, New York University — Very long conduits of melt, (lava tubes, magma conduits, glacial drainage tubes) exist in many locations. An idealized model and its stability is analyzed to answer "how far can the fluid flow and remain liquid"? Laboratory experiments show that when a liquid flows in a pipe with the boundary temperature below freezing, a tubular drainage conduit is surrounded by solidified material. When the flow rate into the pipe is set below a fixed value, the tube freezes shut. As flow rate is gradually changed downward toward the freezing value, pressure change across the pipe rises to a maximum, a result that is not in accord with previous theory. A theoretical model allows for a change in radius in the flow direction (similar to some previous injection molding studies), with a mixed pressure-flux upstream boundary condition. Linear stability analyses of this and a simplified model indicate that: (i) for fixed flux, the tube can be infinitely long with minimum pressure as flux is varied; (ii) for fixed pressure drop across the tube, this minimum determines a maximum length; (iii) for the mixed pressure-flux condition, a stable tube exceeds this length. This is a possible explanation for the previously unexplained experimental pressure maximum near freezing. Therefore, distance traveled by melt within the earth might be very sensitive to the conditions that govern upstream pressure and flow rate.

8:52AM MS.00005 Modelling of Transport and Mixing across Gulf Stream , TATYANA KRASNOPOLSKAYA, VLADIMIR IL'CHENKO, Institute of Hydromechanics NASU, VYACHESLAV MELESHKO, OLENA STETSENKO, Kiev National Taras Shevchenko University — The new mathematical model for a stream function of a meandering jet of Gulf Stream is suggested based upon a modification of the von Kármán vortex street stream function. This stream function allows to approximate experimentally found by Bower main coherent structure elements of Gulf Stream in a coordinate frame moving with a speed of the meander: i) an eastward-propagating meandering jet; ii) regions of fluid recirculation below and above meander crests and troughs; iii) regions of westward-propagating fluid below and above the jet and recirculation regions. The inclusion of eddies above the recirculation regions and the jet enhance transport and mixing across the jet. Calculations show that more than one third of the circular area above hyperbolic points may contain warm fluid from a central area of the jet. To study mixing across the jet we examine deformation of this circular area back in time, so we can determine from which part of the jet that area is composed. Contour line tracking method conserving all topological properties in 2-D flows is used for this procedure.

9:05AM MS.00006 Gravity currents in a stratified ambient fluid , MICHAEL PATTERSON, Yale University & University of Bath, UK, ANDREW ASPDEN, Lawrence Berkeley National Laboratory — Motivated by the previous study of Maxworthy et al. 2002 we revisit the problem in which a lock release gravity current propagates into a stratified ambient fluid. High-resolution three-dimensional numerical simulations based on ILES framework are used in conjunction with a simple box model to develop a greater understanding of the complex interactions between the slumping gravity current and the internal waves that develop. Examination of the energetics of the system are carried out for both sub and super-critical gravity currents. The results show that for the sub-critical gravity currents the energy passed from the gravity current to the wave field and subsequently from the wave field back to the gravity current. Calculations of the energy transfer between the gravity current and the internal wave field will be presented.

9:18AM MS.00007 Fracturing of rocks by ice , IOANNA VLAHOU, M. GRAE WORSTER, DAMTP, University of Cambridge — Frost damage, caused by the freezing of water-saturated media, affects plant roots, pavements and the foundations of buildings, and is a major erosional force in rocks. The process has been studied extensively in the case of soils, and mechanisms such as the formation of ice lenses have been identified. Here, we consider the freezing of water in a three-dimensional cavity in a water-saturated, porous, elastic rock. Initially, the expansion of water as it freezes causes flow away from the solidification front, into the porous rock. The Darcy flow in the porous medium controls the pressure field and therefore the freezing temperature. At later times, disjoining thermomolecular forces create a pre-melted film of water between the ice and the rock and cause flow of pore water from the surrounding rock into the cavity. We find that the disjoining forces between the ice and the rock have the dominant effect, so we focus on those later times when the cavity is ice-filled. We solve the coupled set of integro-differential equations governing the elastic stress in the rock and the flow through its pores to determine the evolution of the shape and extent of the ice-filled cavity.

9:31AM MS.00008 Non-Boussinesq axisymmetric gravity currents at high Re , MARIUS UNGARISH, Technion, Haifa, Israel — The propagation of a non-Boussinesq gravity current in an axisymmetric configuration (full cylinder or wedge) is considered. The current of density ρ_c is released from rest from a lock of radius r_0 and height h_0 into an ambient fluid of density ρ_a in a container of height H, adjacent to the horizontal boundary on which propagation occurs. When the Reynolds number Re is large, the resulting flow is governed by the parameters ρ_c/ρ_a and $H^* = H/h_0$. We show that the shallow-water one layer model, carefully combined with a Benjamin-type front condition, provides a versatile formulation for the thickness and speed of the current, for a wide range of the parameters, without any adjustable constants. (The Boussinesq currents are just a small subdomain about $\rho_c/\rho_a = 1$ of this solution). Comparisons with Navier-Stokes solutions and differences with the 2D rectangular counterpart (Ungarish, J. Fluid Mech. 579, 373-382, 2007) are discussed.

9:44AM MS.00009 A numerical study of the energy flux in internal bores , ZACHARY BORDEN, ECKART 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 globally, does a much better job but requires for closure an assumption about the loss of energy across a bore. It is widely agreed that assuming all the energy loss occurs entirely in one of the layers puts bounds on a bore's propagation speed. However, under some circumstances, both assumptions over-predict the propagation speed, implying an energy gain in the expanding layer. We directly examine the flux of energy within internal bores using 2D direct numerical simulations. We find that although there is a global loss of energy across the bore, there is a transfer of energy from the contracting to the expanding layer causing a net energy gain in that layer. The transfer is largely the result of the horizontal pressure gradient caused by a difference in hydrostatic pressure across the bore.

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MT Suspensions II 200H

8:00AM MT.00001 Effects of inertia and viscoelasticity on the orientation dynamics of axisymmetric particles , VIVEKANAND DABADE, GANESH SUBRAMANIAN, EMU, JNCASR, Bangalore-560064,Karnataka — The talk will focus on the analytical investigation of the effects of weak inertia and fluid viscoelasticity on the orientation dynamics of a spheroidal particle in two canonical flow situations: 1. A spheroid sedimenting in a quiescent fluid, and 2. A neutrally buoyant spheroid rotating in a simple shear flow. The spheroidal geometry is taken as representative of the general effects of particle anisotropy in disperse multiphase systems. The orientation distribution of a non-Brownian spheroidal particle remains indeterminate in both sedimentation and shear flow in the Stokes limit. Either of inertia or viscoelasticity remove this indeterminacy. The above problems are analyzed using a novel approach based on the formalism of vectorial spheroidal harmonics together with the generalized reciprocal theorem. We obtain closed-form expressions for the *O(Re)* inertial and *O(De)* viscoelastic torques in sedimentation, and the *O(Re)* angular velocity in simple shear flow, valid for an arbitrary aspect ratio. The present results highlight errors in earlier theoretical and numerical calculations.

8:13AM MT.00002 Particle acceleration model for gas–solid suspensions at moderate Reynolds numbers , SUDHEER TENNETI, RAHUL GARG, Iowa State University, CHRISTINE HRENYA, University of Colorado, RODNEY FOX, SHANKAR SUBRAMANIAM, Iowa State University — Particle granular temperature plays an important role in the prediction of core annular structure in riser flows. The covariance of fluctuating particle acceleration and fluctuating particle velocity governs the evolution of the granular temperature in homogeneous suspensions undergoing elastic collisions. Koch and co–workers (Phys. Fluid. 1990, JFM 1999) showed that the granular temperature has a source term due to hydrodynamic interactions in gas–solid suspensions in the Stokes flow regime. We performed direct numerical simulations (DNS) of freely evolving suspensions at moderate Reynolds numbers using the immersed boundary method (IBM). We found that simple extension of a class of mean particle acceleration models to their instantaneous counterparts does not predict the correct fluctuating particle acceleration–fluctuating velocity covariance that is obtained from DNS. The fluctuating particle velocity autocorrelation function decay and the Lagrangian structure function obtained from DNS motivate the use of a Langevin model for the instantaneous particle acceleration.

8:26AM MT.00003 Batch Sedimentation in Impulsively Heated nearly mono-disperse Viscous Multiphase Fluids[1] , AMEYA JOSHI, THOMAS WARD, North Carolina State University — Suspensions of nearly mono-disperse spheres, subjected to impulsive-constant temperature heating from below, are studied in a batch sedimentation process. Experiments are performed on suspensions with a range of concentrations and different temperatures, to analyze the effects of varying temperatures on the rate of settling, shock formation and shock velocities. CCD imaging is used to observe and study the settling phenomenon. The Kynch theory of sedimentation (Trans. of Far. Soc., 1952) explains shock formation in low concentrations suspensions and states that particle speed is completely determined by the local density only. We extend this study to include both heating and high concentration suspensions. Higher concentration suspensions (greater than 20%) are characterized by a single shock. Two distinct shocks are observed for all the experiments involving heating from bottom with an exception for the non-heated setup where only one shock is observed. Vortex patterns observed for the heated and the unheated case also show a substantial difference. The distinct vortex patterns produced in the suspension during settling are attributed to heating. The experiments suggest that other models may be more appropriate for high concentration studies.

[1]NC Space Grant

8:39AM MT.00004 Shear-induced fluid tracer diffusion in a semi-dilute suspension of spheres , TAKUJI ISHIKAWA, TAKAMI YAMAGUCHI, Tohoku University — We calculated tracer diffusion in a sheared suspension of non- Brownian rigid spheres and propose a new numerical method based on a boundary element method and Stokesian dynamics method. We present details of the numerical method and examine the accuracy of the method. The limitation of semi-diluteness is due to the accuracy of tracer velocity calculation. The results show that the diffusivity of fluid tracers is greater than that of suspended spheres in the semi-dilute regime. The diffusivity of velocity gradient direction is about threefold greater than that in the vorticity direction. Simple scaling demonstrates that the diffusivity of fluid tracers increases with the square of the volume fraction of spheres in the semi-dilute regime, which is confirmed numerically.

8:52AM MT.00005 Yielding and vorticity banding in sheared nanotube suspensions , ERIK K. HOBBIE, Department of Physics, Department of Coatings and Polymeric Materials, North Dakota State University — We analyze the yielding and flow of nanotube suspensions in the parameter space of particle concentration, aspect ratio and applied shear stress. The density of mechanical entanglements increases with both the aspect ratio and number density of nanotubes, creating an elastic particle network at very low nanotube concentration. Under simple shear flow, these networks exhibit a well-defined yield stress and classic Bingham-fluid behavior. Unstable flows at moderate Peclet number are characterized by a periodic pattern of vorticity-banded domains, which transition to a stable and homogeneous flow-aligned state in the limit of very large Peclet number.

9:05AM MT.00006 Numerical simulation of lateral segregation during Stokes bidisperse sedimentation and onset of irreversibility , XIAOLONG YIN, Assistant Professor, Petroleum Engineering, Colorado School of Mines — It is known that in Stokes bidisperse sedimentation particles of different sizes or densities will segregate in the lateral direction and form column structures. Numerical simulations were used to study lateral segregation in periodic domains containing 1146 particles of same size but different densities (particle-fluid density ratios = 1.4 and 0.6). The volume fractions were 0.15 and 0.15. The presence of lateral segregation was confirmed using a combination of visual observation and measurements of sedimentation velocities, velocity variance, and structure factors. By changing the direction of gravity during sedimentation, the reversibility of the suspension was tested. It was found that the system demonstrated reversibility for only about 6 Stokes times, and then lost its memory. Simulations conducted in domains with top and bottom walls did not show significant lateral segregation. The hindered settling velocities of the suspension in wall-bounded domains were between the predictions of Davis and Gecol (AIChE J. 40:570 1994) and those of Revay and Higdon (J. Fluid Mech. 243:15 1992).

9:18AM MT.00007 Self-organization of fiber suspension , ALEXANDRE FRANCESCHINI, NYU, Dept Phys, Ctr Soft Matter Res, New York, NY 10003 USA, ELIZABETH GUAZZELLI, Aix Marseille Univ U1, IUSTI, CNRS, UMR Polytech Marseille 6595, F-13453 Marseille 13, France, DAVID PINE, NYU, Dept Phys, Ctr Soft Matter Res, New York, NY 10003 USA — A single buoyant non-Brownian fiber in a low Reynolds shear flow has a fully determined motion, so-called Jeffery orbit. However, the behavior of a concentrated fiber suspension remains unclear; even slight interactions between objects can disturb the orbits and collective behavior cannot be neglected [1]. By observing the system under a periodic shear, we are able to specifically observe the irreversible motions due to fiber-fiber interactions. collective motion is followed by viscosity measurements and quantitative image analysis [2, 3]. The angle and center-of-mass of marked fibers are also tracked. The system self-organizes and always becomes more reversible with time until a steady state, fluctuating or not, is reached. A dynamical phase transition between a quasi-reversible and a fluctuating state is observed, with a transient time that exhibit a power law divergence at the critical points. [1] Okagawa A and Mason SG, Science, Volume 181, Issue 4095, p159 (1973) [2] Pine DJ and al., Nature, vol 438, Issue 7070, p997 (2005) [3] Corte L and al., Nature physics, vol 4, Issue 5, p420 (2008)

9:31AM MT.00008 Guided colloidal crystallization in a galvanic micro reactor , CHRISTIAN PUNCKT, LINDA JAN, Princeton University, Princeton, NJ 08544, BORIS KHUSID, New Jersey Institute of Technology, University Heights, Newark, NJ 07102, ILHAN A. AKSAY, Princeton University, Princeton, NJ 08544 — We present a novel method for assembling colloidal particles into an ordered coplanar array of two-dimensional crystals. This technique utilizes an autonomous galvanic micro reactor to control the location and morphology of colloidal crystals. Coplanar arrays of copper and gold microelectrodes are placed into a dilute water solution of hydrochloric acid to form a galvanic couple between the copper acting as an anode and the gold as a cathode. Under appropriate conditions, colloidal particles suspended in the solution assemble into two-dimensional colloidal crystals adherent to the anodic copper. Polystyrene and silica particles having similar sizes and zeta potentials but different densities were employed to study the fluid flow in the galvanic reactor. Spatially resolved, optical analysis of the reaction rate was used to estimate the magnitude and distribution of the electric current over the copper electrodes. Physical mechanisms governing the particle motion and aggregation will be discussed.

9:44AM MT.00009 Order transition in non-colloidal Couette suspension flows:effects of external torques , KYONGMIN YEO, MARTIN MAXEY, Division of Applied Mathematics, Brown University — Suspensions of non-colloidal particles in Couette flows are investigated by using the force-coupling method. It is observed that a hexagonal order begins to develop near the wall at a volume fraction as low as $\phi = 0.48$, while the suspensions in the center of the channel remains disordered. The ordering transition depends on the ratio of the channel width to the particle radius. It is shown that the order state can be modified by applying external torques on the particles. The hexagonal order of the particles is weakened by the negative torque, leading to the increase of the shear viscosity. The positive torque has a favorable effect on the ordered state. However, if the magnitude of the positive torque exceeds a certain threshold, the hexagonal order begins to be weakened. On the other hand, at a moderate volume fraction ($\phi = 0.40$), the external torques do not have a significant effect on the suspension rheology. The nonlinear responses of rheological parameters, such as the shear and vortex viscosities, to the external torques are investigated.

9:57AM MT.00010 Flow Irreversibility in Particle Suspensions with Non-Uniform Strain[1] , J.S. GUASTO, A.S. ROSS, Haverford College, J.P. GOLLUB, Haverford College, University of Pennsylvania — Sheared particle suspensions are irreversible even in Stokes flows.[2,3] Here, we extend this previous work to a system with spatially varying strain, an oscillatory, rectangular channel flow. The particle volume fraction is 40% and the Reynolds number is $\sim 10^{-3}$. Deviations from reversible behavior are parameterized by the mean square particle displacements sampled once per cycle, and are compared to the local strain obtained from measured velocity profiles. Strikingly, the particle motions across the entire channel become irreversible simultaneously, despite the non-uniform local strain, and the irreversibility increases roughly exponentially with the wall strain. Measured velocity profiles are plug-like near the channel center, which may indicate shear-induced particle migration. We examine the possibility that irreversibility is a mechanism for shear-induced migration.

[1]Supported by NSF DMR-0803153.

[2]D.J. Pine, *et al.*, Nature **438**, 997 (2005).

[3]L. Corté, *et al.*, Nature Physics **4**, 420 (2008).

Tuesday, November 24, 2009 8:00AM - 9:57AM –

Session MU Separated Flows II and Miscellaneous Experiments 200I

8:00AM MU.00001 Control of Flow past a Stationary Circular Cylinder: Effects of a Single Straight Wire and a System of Helical Wires , ALIS EKMEKCI, University of Toronto, DONALD ROCKWELL, Lehigh University — Effects of geometric disturbances on the surface of a circular cylinder are investigated experimentally via Particle Image Velocimetry (PIV). Consideration is given to a single straight wire and a system of three equidistant helical wires. The Reynolds number is 10,000 and the scale of the wire is about one percent of the cylinder diameter; moreover, the wire diameter is smaller than the thickness of the unperturbed boundary layer developing around the cylinder. A straight wire, located at an angle of 60° from the forward stagnation point of the cylinder, yields bistable shear layer oscillations between two distinct states involving reattachment and no reattachment downstream of separation at the wire. For the helical wire system, bistable oscillations also occur at the flow cross-section where the helical wire passes through this same critical angular location on the cylinder surface. These types of small-scale geometric disturbances do not attenuate the Karman instability in the presence of bistable oscillations. However, they markedly influence the onset of instability of the separated shear layer.

8:13AM MU.00002 Manipulating the Forces on a Sphere Using a Dynamic Roughness Element[1]

, A.K. NORMAN, B.J. MCKEON, California Institute of Technology — Though the effect of distributed roughness on flow over a sphere has been examined in detail, there have been few observations as to the effect of an isolated roughness element on the forces induced on a sphere that is in uniform flow. In this experimental study, we examine how the forces are altered due to both a stationary and dynamic three-dimensional roughness element in the Reynolds number range of 5×10^4 to 5×10^5. It is found that even a small change to the geometry of the sphere, by adding a cylindrical roughness element with a width and height of 1% the sphere diameter, dramatically alters the drag and lateral forces over a wide range of Reynolds numbers. Of particular interest is that the mean of the lateral force magnitude can be increased by a factor of about seven, compared with a stationary roughness element, by moving the isolated roughness at a constant angular velocity about the sphere. The interaction of the roughness element with the flow is examined to understand the cause of the large forces.

[1]Support from the National Science Foundation under Grant No. 0747672 (Program Manager William W. Schultz) is gratefully acknowledged.

8:26AM MU.00003 Control of the Near-Wake of a Forced-Oscillating Cylinder via a Straight Surface Wire

, TAYFUN AYDIN, VEITCH THOMAS, ALIS EKMEKCI, University of Toronto — Effects of a spanwise surface wire on the near-wake of a circular cylinder subjected to small-amplitude, forced- oscillation at the inherent Karman frequency are studied via Particle Image Velocimetry (PIV). The Reynolds number is 10,000 and the diameter of the surface wire is two orders of magnitude smaller than the cylinder diameter, but larger than the unperturbed boundary layer thickness prior to its separation. The near wake is markedly affected by the wire when it is located within a certain range of angular positions with respect to the approach flow. As the angular position is altered, the near wake undergoes a contraction, followed by an extension in the streamwise direction. This trend is in the reverse order of what has been reported for a stationary cylinder. The predominant frequency of the velocity fluctuations in the separating shear layers locks onto the frequency of the forced oscillation and remains unchanged with the angular position of the wire. However, the predominant frequency in the near wake shows a gradual decrease, then increase, as the angular position of the wire is increased within the defined range of angles.

8:39AM MU.00004 An Experimental Study of Flow Around a Spinning Cylinder[1]

, CESAR CANTU, BENITO GONZALEZ, SANJAY KUMAR, Department of Engineering, The University of Texas at Brownsville, TX-78520 — Flow around a single spinning circular cylinder is studied experimentally. The experiments are carried out in a free-surface water tunnel at Reynolds numbers (*Re*) of 200, 300, and 400 and non-dimensional rotation rates (ratio of surface speed of cylinder to free stream velocity), α, varying from 0 to 5. The diagnostics was done by flow visualization using hydrogen bubble technique. We present the global view of the wake structure at the three Reynolds numbers and various rotation rates. Vortex shedding suppression is observed for $\alpha \sim$2.0. Experimental evidence of the vortex shedding in the second vortex shedding regime (4.34$< \alpha <$4.70) (S. Mittal and B. Kumar, J.Fluid Mech., 2003) is presented for the first time at *Re*=200. Strouhal number (*St*) measurements and global wake patterns agree well with the computations of Mittal and Kumar (2003) at Reynolds number of 200. Strouhal number measurements in the Reynolds number range and rotation rates indicate that at low values of α ($\sim \alpha <$1.2) *St* increases with *Re* while for larger values of α ($\sim$1.2$< \alpha <$2.0), *St* decreases with *Re*.

[1]The support from the grant NSF MRI-CMMI-0723094 is acknowledged.

8:52AM MU.00005 Analysis of the breakup of a viscous thread in a turbulent flow

, MONICA MARTINEZ-ORTIZ, ERNESTO MANCILLA, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — An experimental study was conducted to analyze the stability and breakup of a viscous thread in an isotropic turbulent flow. The motivation for this study arises from the need to understand the mechanisms that control the formation of emulsions of very viscous liquids. Experiments were performed in an isotropic turbulence chamber, in which a single thread was injected. The fluid disturbances on the thread's surface were studied for filaments of different diameters and lengths. The turbulence intensity was varied for each case. The fluid velocity was characterized using a Particle Image Velocimetry (PIV) system. The threads and their temporal evolution were visualized with a high speed camera. We observed that for most conditions the filaments are surprisingly stable; they are largely elongated until their diameter is very small. A dimensionless analysis indicated that at large diameters the filaments respond to turbulent fluctuation, while at small diameters capillary forces dominate.

9:05AM MU.00006 Grid turbulence study using particle image velocimetry

, J.I. CARDESA-DUENAS, T.B. NICKELS, University of Cambridge — We analyse grid turbulence using two-dimensional Particle Image Velocimetry (PIV) at moderate Reynolds numbers ($Re_M = 16000$). Measurements on two orthogonal planes are taken at several downstream locations in the mature turbulence region. The resolution is within three times the Kolmogorov length scale. This is made possible by working with solid particles in water, where denser seedings can be achieved than with oil droplets in wind tunnels as previously attempted. We deduce four derivative moments which are sufficient to measure dissipation and mean square vorticity assuming axisymmetric turbulence, a far less restrictive assumption than isotropy. The disspation estimate is compared with that found from the turbulent kinetic energy balance. The tendency towards isotropy from the axisymmetric state is discussed in the light of four determining parameters (Batchelor 1946). Measurements are also taken immediately after the grid, where the vortical structures shed by the grid can be visualised and tracked as they evolve downstream.

9:18AM MU.00007 Properties of the Measured Instantaneous Viscous Boundary Layer Thickness in Turbulent Rayleigh-Benard Convection.[1]

, KE-QING XIA, QUAN ZHOU, Department of Physics, The Chinese University of Hong Kong, Shatin, Hong Kong, China — We report measurements of the instantaneous viscous boundary layer (BL) thickness δ_v(t) in turbulent Rayleigh-Benard convection. The instantaneous δ_v(t) obtained from PIV-measured two-dimensional velocity field is found to exhibit intermittent fluctuations. It is also found that there is a clean separation of statistical behavior of δ_v(t) below and above its most probable value δ_v^{mp}: for δ_v(t)$< \delta_v^{mp}$, it obeys a lognormal distribution and for δ_v(t)$> \delta_v^{mp}$, the distribution of δ_v(t) has an exponential tail. Our results reveal that the variation of δ_v(t) responds negatively to the fluctuations of the large-scale mean flow velocity with a nonzero time delay; while close to the plate the horizontal velocity reflects the variation of δ_v(t) with zero time delay. In the reference frame of the time-dependent thickness δ_v(t), the conditional-averaged velocity profile agrees excellently with the theoretical Prandtl-Blasius laminar BL profile and the shape factor is found to be much closer to the theoretical value of the Blasius shear layer as compared to that obtained in the laboratory frame. It is further found that δ_v^{mp} scales as Re$^{0.5}$.

[1]Work supported by the Research Grants Council of Hong Kong SAR (Project Nos. CUHK 403806 and 403807).

9:31AM MU.00008 Time-dependent ventilation flows driven by opposing wind and buoyancy

, I. COOMARASWAMY, C.P. CAULFIELD, BPI & DAMTP, University of Cambridge — We consider an enclosure containing an isolated heat source, ventilated by a windward high level opening and a leeward low level opening, so that prevailing wind acts to oppose buoyancy driven flow. By conducting dynamically similar salt bath experiments in a recirculating flume tank (Hunt & Linden, *J. Fluid Mech.* **527**, 27 (2005).), we investigate the initial value problem of "box filling" with constant opposing wind for a number of different opening sizes and wind strengths. We employ a novel method of flow visualisation (Dalziel, Patterson, Caulfield & Coomaraswamy, *Phys. Fluids* **20**, 065106 (2008).), backlighting apparatus with a panel of electroluminescent tape and employing dye attenuation techniques, allowing us to track the evolution of the stratification within the interior. Our findings demonstrate that some unusual transient phenomena can occur, as predicted by theoretical models we have previously developed for the system. We evaluate the accuracy of these models with regard to the types of transient and final states seen for each set of conditions, and also use our experimental data for the interior density distribution to examine the validity of the models' underlying assumptions.

9:44AM MU.00009 Various methods for extracting forces on a moving plate using Time-Resolved Particle Image Velocimetry (TR-PIV) , D. RIVAL, D. SCHOENWEITZ, C. TROPEA, Technische Universitaet Darmstadt, Darmstadt, Germany — It is often very challenging or even impossible to measure the forces directly on swimming or flying animals. For this reason traditional control-volume methods are used to estimate the unsteady forces on the body in question. TR-PIV systems allow for the measurement of the full spatial and temporal velocity field in the control volume as well as along the bounding control surfaces. The corresponding pressure field can be integrated from the complete velocity-field data set. However, the measurement of the velocity field in the proximity of the bodys surface is at times cumbersome due to shadows and reflections. An alternate control-volume formulation[1] eliminates the need for velocity measurements in the proximity of the body. This method has been tested and compared to the traditional control-volume technique for a generic pitching and plunging flat plate in a hover chamber. The advantages and disadvantages of these methods are discussed in the context of their measurement accuracy.

[1]J. Z. Wu, Z.-L. Pan and X.-Y. Lu, "Unsteady fluid-dynamic force solely in terms of control-surface integral, Phys. of Fluids **17**, 098102 (2005)

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MV Swimming V: Micro-organisms II 205A-D

8:00AM MV.00001 The combined effect of gravity and stresslets on the instability of a uniform suspension of swimming micro-organisms , TIMOTHY PEDLEY, University of Cambridge — Uniform dilute suspensions of gyrotactic, swimming micro-organisms (bottom-heavy algae) become unstable because of the effect of gravity [1]; suspensions of head-heavy organisms are stable. In the absence of gravity, uniform suspensions of aligned swimmers become unstable because of the stresslet distribution generated by their swimming actions, whether they are pullers (algae) or pushers (bacteria) [2,3], but only pushers cause instability of an isotropic suspension [3]. Here we examine the effect of weak gravity on a suspension's instability, and find that even a small gravitational term leads to instability for bottom-heavy cells at small enough wavenumber, whatever the magnitude and sign of the stresslet term, but may not be enough to stabilise suspensions of head-heavy pushers at a high enough number density of cells. However, use of realistic parameter values suggests that gravity will normally be dominant.

[1] T J Pedley & J O Kessler, Ann. Rev. Fluid Mech. **24**,313 (1992)
[2] R A Simha & S Ramaswamy, Phys. Rev. Lett. **89**,058101 (2002)
[3] D Saintillan & M J Shelley, Phys. Fluids **20**,123304 (2008)

8:13AM MV.00002 The dilute rheology of swimming suspensions: A simple kinetic model , DAVID SAINTILLAN, MechSE, University of Illinois at Urbana-Champaign — A simple kinetic model is presented for the shear rheology of a dilute suspension of particles swimming at low Reynolds number. If interparticle hydrodynamic interactions are neglected, the configuration of the suspension is characterized by the particle orientation distribution, which satisfies a Fokker-Planck equation including the effects of the external shear flow, rotary diffusion, and particle tumbling. The orientation distribution then determines the leading-order term in the particle extra stress in the suspension, which can be evaluated based on the classic theory of Hinch and Leal [J. Fluid. Mech. 52(4):683-712 (1972)], and involves an additional contribution arising from the permanent force dipole exerted by the particles as they propel themselves through the fluid. Numerical solutions of the steady-state Fokker-Planck equation were obtained using a spectral method, and results are reported for the shear viscosity and normal stress differences in terms of flow strength, rotary diffusivity, and correlation time for tumbling. It is found that the rheology is characterized by much stronger normal stress differences than for passive suspensions, and that tail-actuated swimmers result in a strong decrease in the effective shear viscosity of the fluid.

8:26AM MV.00003 Three-Dimensional Pattern Formation in Flowing Suspensions of Swimming Particles , AMIR ALIZADEH PAHLAVAN, DAVID SAINTILLAN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Suspensions of self-propelled particles, such as swimming micro-organisms, are known to undergo complex dynamics as a result of hydrodynamic interactions. Using the kinetic theory recently developed by Saintillan and Shelley ("Instabilities, pattern formation, and mixing in active suspensions", *Physics of Fluids* **20**, 123304, 2008), we investigate the three-dimensional pattern formation occurring in suspensions of active particles. Our numerical simulations confirm the results of the linear stability analysis of Saintillan et al., and the long-time nonlinear behavior is shown to be characterized by the formation of strong density fluctuations, which merge and break up in time in a quasiperiodic fashion. These complex motions result in very efficient fluid mixing, which we quantify by means of a multiscale mixing norm. The effects of an external shear flow on the pattern formation are also investigated using both simulations and a stability analysis.

8:39AM MV.00004 Linear Stability for Models of Active Suspensions , CHRISTEL HOHENEGGER, MICHAEL SHELLEY, Courant Institute New York University — Recent work by Saintillan & Shelley has modeled the dynamics of active suspensions, such as swirling bacterial baths, through a modification of Doi rod theory. However, the sign of the dipolar extra stress can be of the opposite sign of Doi theory, and this leads to large-scale flow instability. We investigate the structure of this system linearized near a state of isotropy and uniformity. We show that concentration fluctuations generally decay, and that while long-wave instability depends upon the particular swimming mechanism, short-wave stability does not. This suggests that the model is well-posed, even in the absence of translational and rotational diffusion effects.

8:52AM MV.00005 Colloidal motility and patterning by physical chemotaxis , JEREMIE PALACCI, Universite de Lyon, BENJAMIN ABECASSIS, LETI, CEA Grenoble, CECILE COTTIN-BIZONNE, CHRISTOPHE YBERT, LYDERIC BOCQUET, Universite de Lyon — We developped a microfluidic setup to show the motility of colloids or biomolecules under a controlled salt gradient thanks to the diffusiophoresis phenomenon [1,2]. We can therefore mimic chemotaxis on simple physical basis with thrilling analogies with the biological chemotaxis of E. Coli bacteria: salt dependance of the velocity [3] and log-sensing behavior [4]. In addition with a temporally tunable gradient we show we can generate an effective osmotic potential to trap colloids or DNA. These experimental observations are supported by numerical simulations and an asymptotic ratchet model. Finally, we use these traps to generate various patterns and because concentration gradients are ubiquitous in nature, we question for the role of such a mecanism in morphogenesis [5] or positioning perspectives in cells [6].

[1] B. Abecassis, C. Cottin-Bizonne, C. Ybert, A. Ajdari, and L. Bocquet, Nat. Mat., 7(10):785–789, 2008. [2] Anderson, Ann. Rev. Fluid Mech, 21, 1989. [3] Y. L. Qi and J. Adler, PNAS, 86(21):8358–8362, 1989. [4] Y. V. Kalinin, L. L. Jiang, Y. H. Tu, and M. M. Wu, Biophys. J., 96(6):2439–2448, 2009. [4] J. B. Moseley, A. Mayeux, A. Paoletti, and P. Nurse, Nat., 459(7248):857–U8, 2009. [6] L. Wolpert, Dev., 107:3–12, 1989

9:05AM MV.00006 Bacterial streamers in curved microchannels[1] , ROBERTO RUSCONI, SIGOLENE LECUYER, Harvard University, LAURA GUGLIELMINI, HOWARD STONE, Princeton University — Biofilms, generally identified as microbial communities embedded in a self-produced matrix of extracellular polymeric substances, are involved in a wide variety of health-related problems ranging from implant-associated infections to disease transmissions and dental plaque. The usual picture of these bacterial films is that they grow and develop on surfaces. However, suspended biofilm structures, or streamers, have been found in natural environments (e.g., rivers, acid mines, hydrothermal hot springs) and are always suggested to stem from a turbulent flow. We report the formation of bacterial streamers in curved microfluidic channels. By using confocal laser microscopy we are able to directly image and characterize the spatial and temporal evolution of these filamentous structures. Such streamers, which always connect the inner corners of opposite sides of the channel, are always located in the middle plane. Numerical simulations of the flow provide evidences for an underlying hydrodynamic mechanism behind the formation of the streamers.

[1]This work was supported by BASF via the BASF Advanced Research Initiative at Harvard University.

9:18AM MV.00007 Swarming dynamics in bacterial colonies , HEPENG ZHANG, AVRAHAM BE'ER, RACHEL SMITH, E.-L. FLORIN, HARRY L. SWINNEY, Univ. of Texas at Austin — Swarming is a widespread phenomenon observed in both biological and non-biological systems. Large mammal herds, fish schools, and bird flocks are among the most spectacular examples. Many theoretical and numerical efforts have been made to unveil the general principles of the phenomenon, but systematic experimental studies have been very limited. We determine the characteristic velocity, length, and time scales for bacterial motion in swarming colonies of *Paenibacillus dendritiformis* growing on semi-solid agar substrates. The bacteria swim within a thin fluid layer, and they form long-lived jets and vortices. These coherent structures lead to anisotropy in velocity spatial correlations and to a two-step relaxation in velocity temporal correlations. The mean squared displacement of passive tracers exhibits a short-time regime with nearly ballistic transport and a diffusive long-time regime. We find that various definitions of the correlation length all lead to length scales that are, surprisingly, essentially independent of the mean bacterial speed, while the correlation time is linearly proportional to the ratio of the correlation length to the mean speed.

9:31AM MV.00008 Dynamics of Swimming Particles in Chaotic Fluid Flows , NIDHI KHURANA, JERZY BLAWZDZIEWICZ, NICHOLAS T. OUELLETTE, Yale University — We numerically investigate the effect of swimming (modeled as an intrinsic velocity) on the transport of particles in chaotic two-dimensional, incompressible fluid flows. We consider spheroidal particles advected by an externally imposed flow, and show that even a small amount of swimming significantly changes the particle dynamics. The phase space is examined as the magnitude and direction of the swimming speed are varied.

9:44AM MV.00009 Turbulence and motility conspire to generate small-scale phytoplankton patchiness[1] , ROMAN STOCKER, MIT, ERIC CLIMENT, Institut de Mecanique des Fluides de Toulouse, WILLIAM DURHAM, MIT — Phytoplankton are heterogeneously distributed at nearly all scales in the Ocean. Small-scale patchiness is often thought to result from a passive "top down" process where large patches of phytoplankton are broken into subsequently smaller ones by turbulent motion. Here we demonstrate instead a "bottom up" process in which patchiness is generated *ex novo* by the coupling of turbulence and cell motility. We implemented an individual based model (IBM) for phytoplankton cells within a direct numerical simulation (DNS) of turbulence. The IBM describes the movement behavior of gyrotactic cells, for which the swimming direction is dictated by the interaction of cell morphology (e.g. bottom-heaviness) and fluid shear. Two dimensionless parameters govern the dynamics: the ratio of the Kolmogorov velocity to swimming velocity, and the ratio of Kolmogorov time scale to cell reorientation time. We find that this mechanism can indeed generate strong patchiness and discuss the parameter regime under which this occurs. An analytical single-vortex model helps to shed light on the fundamental physics at play. These findings strongly highlight the importance of microbial motility in the Ocean.

[1]Partially funded by NSF.

9:57AM MV.00010 Motility-enhanced bioflocculation , MITUL LUHAR, Massachusetts Institute of Technology, ZHAOXUAN ZHANG, National University of Singapore, ROMAN STOCKER, Massachusetts Institute of Technology — Bacteria often rely on their ability to aggregate to survive in nature. They can form clusters among themselves or with suspended colloids, leading to biologically-enhanced flocculation (bioflocculation). We investigate the role of cell motility on bioflocculation by comparing two strains of *Escherichia coli,* a wild type and a non-motile mutant. We quantify settling rates from a suspension of bacteria and 1 μm polystyrene beads, by independently varying the concentration of bacteria and colloids. We find that motility enhances settling rates up to 6-fold**.** We rationalize our findings in terms of an increase in encounter rates between bacteria and colloids. These results could contribute to improve the performance of wastewater treatment processes and provide a possible explanation for why motile bacteria are more successful biofilm formers.

Tuesday, November 24, 2009 8:00AM - 10:10AM –
Session MW Micro Propulsion I 208A-D

8:00AM MW.00001 Swimming in gels , HENRY FU, Brown University, CHARLES WOLGEMUTH, University of Connecticut Health Center, VIVEK SHENOY, THOMAS POWERS, Brown University — Many swimming microorganisms must move through viscoelastic fluids and gels. In this talk I focus on swimming through gels. First, unlike incompressible fluids, a gel can have compressional modes with relative motion between polymer and solvent fractions. In a continuum model for a gel, we show that compressibility can increase the swimming speed of Taylor's swimming sheet. The zero-frequency shear modulus of a gel requires altered boundary conditions on the swimmer. Second, many biological gels are heterogeneous on the lengthscale of swimming microorganisms, necessitating non-continuum models that treat the gel network and swimmer on equal footing. We show that a random network modeled as dilute, immobile spherical obstacles increases the average swimming speed of a Golestanian three-sphere swimmer.

8:13AM MW.00002 Modeling magnetically driven synthetic microcapsules , HASSAN MASOUD, ALEXANDER ALEXEEV, Georgia Institute of Technology — Using computer simulations and theory, we examine how to design magnetically-responsive synthetic microcapsules that able to move in a steady manner in microfluidic channels. These compliant fluid-filled capsules encompass superparamagnetic nanoparticles in their solid shells and, thereby, can be manipulated by alternating magnetic forces. To model the magnetic capsules propelled in fluid-filled microchannels, we employ a hybrid computational method for fluid-structure interactions. This method integrates the lattice Boltzmann model for the fluid dynamics and the lattice spring model for the micromechanics of solids. We show that in circulating magnetic field the capsules propel along sticky microchannel walls. The direction of capsule motion depends on the relative location of the solid surface, whereas the propulsion speed can be regulated through the wall adhesiveness, amplitude and frequency of magnetic forces, and elasticity of capsule's shell. The results indicate that such mobile fluid-filled containers could find application in lab-on-chip systems for controlled delivery of minute amounts of fluidic samples.

8:26AM MW.00003 Motion of microbeads propelled by bacterial chemotaxis in a microfluidic platform[1] , DONG WOOK KIM, YOUNG WON KIM, JUNG YUL YOO, Department of Mechanical and Aerospace Engineering, Seoul National University — Micro actuators propelled by bacteria are of great interest in recent years, because bacterial chemotaxis has well-presented one of the very promising solutions to the utilization of the motion of flagellated bacteria. In this work, the motion of fluorescent microbeads driven by bacterial chemotaxis has been analyzed by micro-particle tracking velocimetry (μ-PTV). Flagellated bacteria, Serratia Marcescens, are attached to the surface of the polystyrene (PS) microbeads spontaneously in an aqueous culture solution. Then, these particles are injected in the test medium where the linear concentration gradient of L-aspartate is maintained, which is generated by convective and molecular diffusions in a microfluidic platform. It is observed that the particles slowly move toward the high-concentration zone of L-aspartate. This work shows that migration of microbeads using bacterial chemotaxis can be one of the effective tools for the applications to actuators of micro-bio robots.

[1]Supported by the Korea Research Foundation Grant funded by the Korean Government (MOEHRD).

8:39AM MW.00004 Digital holographic PTV analysis of the motile performance of the red-tides alga "*Cochlodinium polykrikoides*"[1] , M. SOHN, S. LEE, Y. CHOI, G. SEO, POSTECH, Y. KANG, Y. KANG, NFRDI, BIOFLUID AND BIOMIMIC RESEARCH CENTER, POSTECH COLLABORATION, MARINE ECOLOGY RESEARCH DIVISION, NFRDI COLLABORATION — The outbreaks of *Cochlodinium polykrikoides* causes severe damages to fisheries in Korea and Japan. *C. polykrikoides* actively makes chains in its exponential and stationary growth phase. The role of the chain formation has not been clearly known yet. In the present study, the motility characteristics of *C. polykrikoides* was investigated using 3D digital holographic PTV technique. The moving trajectories and average velocities of the single and multi-chained *C. polykrikoides* were compared to investigate the effect of the chain formation on the motile performance. Most of the multi-chained *C. polykrikoides* exhibited zigzag or helical trajectory and the translational velocity of the 4-chained *C. polykrikoides* was at least two-times larger than the single one.

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

8:52AM MW.00005 Cilia-induced fluid mixing in mucociliary clearance , SARAH LUKENS, Tulane University, XINGZHOU YANG, Mississippi State University, LISA FAUCI, Tulane University — The surfaces of the upper airways to the lung are coated with a surface liquid (ASL), which prevents inhaled pathogens from accumulating in the lungs. The ASL contains a watery periciliary layer (PCL), where a dense mat of cilia beat, propelling the viscous mucus layer above toward the trachea and mouth. We investigate fluid transport and mixing in the PCL based upon a computational model that couples the internal force generating mechanisms of cilia with external fluid dynamics, including an elastic mucus layer. We use Lagrangian Coherent Structure (LCS) methods to identify different spatial mixing regions, and study qualitative behavior of cilia with and without the mucus layer.

9:05AM MW.00006 Swimming of helically-undulating rings in a Stokes fluid , LISA FAUCI, HOA NGUYEN, RICARDO ORTIZ, RICARDO CORTEZ, Tulane University — Dinoflagellates swim due to the action of two eucaryotic flagella - a trailing, longitundinal flagellum that propagates planar waves, and a transverse flagellum that propagates helical waves. The transverse flagellum wraps around the cell in a plane perpendicular to the trailing flagellum, and is thought to provide both forward thrust along with rotational torque. Motivated by the intriguing function of this transverse flagellum, we study the fundamental fluid dynamics of a helically-undulating ring in a Stokes fluid. We use slender-body theory to compute the steady-state transverse and rotational swimming velocities of the ring in free- space, due to an imposed helical traveling wave. In addition, we study the dynamics of an undulating, elastic ring moving in both free-space and near a plane wall using the method of regularized Stokeslets.

9:18AM MW.00007 A lightning stab in the dark: fluid dynamics of attack jumps of ambush feeding copepods , TOMAS BOHR, Department of Physics and Center for Fluid Dynamics, Technical University of Denmark, ANDERS ANDERSEN, VINCENT LANGLOIS[1], Department of Physics and Center for Fluid Dynamics, Technical University of Denmark, THOMAS KIØRBOE, HANS HENRIK JAKOBSEN, National Institute for Aquatic Resources, Technical University of Denmark — A large class of marine zooplankton, in particular among copepods, are "ambush feeders", who wait for their prey and capture them by surprise attacks. The successful attack must happen so rapidly and unexpectedly that the prey cannot escape and the fluid disturbances created by the attack must be so small that the prey is not pushed away by the flow created by the much larger approaching copepod. Detailed high speed video *in vivo* reveals that the nearly blind copepod manages to perform the attack by precision maneuvering during a rapid jump of a few milliseconds moving approximately one body length. The prey is pushed only by around 10% of this distance, and thus the bulk flow must be close to potential with small boundary layers. From this, we argue that the smallest ambush feeding copepod, *Oithona davisae*, is close to the size limit for the ambush feeding strategy. REF: T. Kiørboe, A. Andersen, V. J. Langlois, H. H. Jakobsen and T. Bohr, PNAS (2009)

[1]Present address: Laboratoire de Sciences de la Terre, Université Claude Bernard Lyon

9:31AM MW.00008 Propulsion of water striders: capillarity and hydrodynamics , PENG GAO, JAMES FENG, Department of Chemical and Biological Engineering, University of British Columbia — We present a numerical investigation of the propulsion of water striders. The flow produced by the stroke of the leg is modeled as that around a hydrophobic circular cylinder astride the interface. We use a diffuse-interface model to compute the moving contact line and the interfacial deformation, and to probe the origin of the propulsive force acting on the leg. The movement of the leg produces a significant deformation of the interface: The upstream meniscus is curved by the dynamic pressure, while the downstream meniscus is flattened by the low-pressure wake associated with vortex shedding. Due to this asymmetry, the force produced by interfacial tension dominates the propulsion.

9:44AM MW.00009 Comparison of real and idealized cetacean flippers[1] , MARK MURRAY, U.S. Naval Academy, PAUL WEBER, LAURENS HOWLE, Duke University, FRANK FISH, West Chester University — We explored the consequences of the idealization process by creating exact scale models of cetacean flippers using CT scans, creating corresponding idealized versions, then determining the hydrodynamic characteristics of the models via water tunnel testing. We found that the majority of the idealized models did not exhibit fluid dynamic properties that were drastically different from those of the real models, although multiple consequences resulting from the idealization process were evident. Drag performance was significantly improved by idealization. Overall, idealization is an excellent way to capture the relevant effects of a phenomena found in nature, which spares the researcher from having to painstakingly create exact models, although we have found that there are situations where idealization may have unintended consequences such as one model that exhibited a decrease in lift performance.

[1]Authors would like to acknowledge support from NSF.

9:57AM MW.00010 The hydrodynamics of two species of copepods: temperate and subtropical *Euchaeta* , D.R. WEBSTER, Georgia Tech, K.B. CATTON, Colorado State Univ., J. YEN, Georgia Tech — Different species of the copepod genera *Euchaeta* live in polar, temperate, and subtropical ocean environments. *Euchaeta elongata* is a species found in temperate waters and is roughly double the size of the subtropical species *Euchaeta rimana*. The kinematic viscosity of the ocean water in the temperate latitude (8 deg C) is roughly 50% greater than that of subtropical environments (23 deg C). We hypothesize that these species have adapted to the local fluid environment to create flow disturbances that facilitates optimal prey capture and predator avoidance. Particle Image Velocimetry (PIV) was used to quantify the flow surrounding each copepod species during cruising and escaping behaviors. Seven to nine replicates for each species were collected for free swimming specimens during both cruise and escape behavior. The average Reynolds number of both species was found to be on the order of 10 for cruising behavior and 1000 for escapes. During cruising, the spatial extent of the region of flow disturbance, defined by a threshold of the maximum principle rate of deformation, was not significantly different between species. In contrast, the spatial extent of the region of flow disturbance during escapes was larger for *E. elongata*. Further, the viscous dissipation rate was similar for the species during cruising, whereas *E. elongata* had a significantly greater viscous dissipation rate during escape behavior.

Tuesday, November 24, 2009 10:30AM - 11:05AM –
Session N1 Invited Session: Patterns and Dynamics in Transitional Shear Flows 205A-D

10:30AM N1.00001 Patterns and dynamics in transitional shear flows , LAURETTE TUCKERMAN, PMMH-ESPCI-CNRS, France — One of the greatest mysteries in fluid dynamics is surely transition to turbulence. The classic shear flows – channel, plane Couette and pipe flow – while linearly stable, undergo sudden transition to 3D turbulence. In recent years, transition has been attacked with an arsenal of weapons from dynamical systems theory, such as low-dimensional chaos, unstable periodic orbits, heteroclinic connections, fractal basin boundaries. At the same time, 3D physical mechanisms such as streamwise vorticity and streaks have supplanted the 2D picture of linear instability long promoted by Squire's theorem. A striking recent discovery by experimentalists at CEA-Saclay is that large-aspect-ratio plane Couette flow near transition actually takes the form of a steady pattern of wide turbulent and laminar bands, with a fixed angle and wavelength. We have been able to reproduce these remarkable flows in numerical simulations of the Navier-Stokes equations. Simulations display a rich variety of variants of these patterns, including spatio-temporal intermittency, branching and travelling states, and localized states analogous to spots. Because similar patterns have since also been observed in Taylor-Couette, channel and pipe flow, it appears that they are inevitable intermediate states on the route from turbulent to laminar flow in large aspect-ratio shear flows. In addition to their intrinsic interest, these patterns provide clues to the transition to turbulence.

Tuesday, November 24, 2009 10:30AM - 11:05AM –
Session N2 Invited Session: Particle Entrainment under Turbulent Flow Conditions 208A-D

10:30AM N2.00001 Particle Entrainment under Turbulent Flow Conditions[1] , PANAYIOTIS DIPLAS, Baker Environmental Hydraulics Laboratory, Civil and Environmental Engineering, Virginia Tech — Erosion, transportation and deposition of sediments and pollutants influence the hydrosphere, pedosphere, biosphere and atmosphere in profound ways. The global amount of sediment eroded annually over the continental surface of the earth via the action of water and wind is estimated to be around 80 billion metric tons, with 20 of them delivered by rivers to the oceans. This redistribution of material over the surface of the earth affects most of its physical, chemical and biological processes in ways that are exceedingly difficult to comprehend. The criterion currently in use for predicting particle entrainment, originally proposed by Shields in 1936, emphasizes the time-averaged boundary shear stress and therefore is incapable of accounting for the fluctuating forces encountered in turbulent flows. A new criterion that was developed recently in an effort to overcome the limitations of the previous approach will be presented. It is hypothesized that not only the magnitude, but also the duration of energetic near bed turbulent events is relevant in predicting grain removal from the bed surface. It is therefore proposed that the product of force and its duration, or impulse, is a more appropriate and universal criterion for identifying conditions suitable for particle dislodgement. Analytical formulation of the problem and experimental data are used to examine the validity of the new criterion.

[1]Financial support for this study was provided by NSF grant EAR-0439663.

Tuesday, November 24, 2009 11:10AM - 11:30AM –
Session O1 Andreas Acrivos Dissertation Award Lecture 205A-D

11:10AM O1.00001 Andreas Acrivos Dissertation Talk: Computational study of fluid particles: dynamics of drops, rheology of emulsions and mechanics of biological cells , XIAOYI LI, United Technologies Research Center — This abstract not available.

Tuesday, November 24, 2009 11:10AM - 11:30AM –
Session O2 Francois Frenkiel Award Lecture 208A-D

11:10AM O2.00001 Francois Frenkiel Award Talk: Fundamental Aspects of Concentration Polarization Arising from Nonuniform Electrokinetic Transport , ADITYA S. KHAIR, TODD M SQUIRES, University of California, Santa Barbara — .

Tuesday, November 24, 2009 11:40AM - 1:37PM –
Session PA Turbulent Boundary Layers: Complex 101A

11:40AM PA.00001 Mean Flow and Wall Shear Measurements in Turbulent Boundary Layers with Various Imposed Complex Pressure Gradients , PAUL ROZIER, RICHARD DUNCAN, HASSAN NAGIB, IIT, Chicago — Mean flow velocity profiles with miniature Pitot probes, and wall-shear stress measurements with oil film interferometry, are used to document the development of high Reynolds number turbulent boundary layers on a flat plate under the influence of four different complex pressure gradients. The National Diagnostic Facility (NDF) is uniquely suited for establishing and documenting such flow conditions to serve as grounds for testing various turbulence models in well-known, two-dimensional boundary layers. Each of the four conditions starts with a zero pressure gradient and returns to it again along the test section. Two of the conditions include an adverse gradient region followed by a favorable one, and the other two cases experience the same two effects in the opposite sequence but to comparable magnitudes of free-stream velocity variations. The mean velocity profiles are measured and used to determine parameters such as the shape factor, the logarithmic overlap-region parameters, and the wake or outer flow parameters. The effect of the initial boundary layer conditions on the development of such complex flows is also examined by changing the position of the pressure gradient "hump" between two sets of two opposite gradients conditions. The results will be briefly compared to computations at IIT of nearly the same flow conditions.

11:53AM PA.00002 Computations of Turbulent Boundary Layers Subjected to Various Localized Pressure Gradients , RICARDO VINUESA MOTILVA, HASSAN NAGIB, IIT, Chicago — Four different localized pressure gradient configurations were computed using a commercially available code by means of four RANS turbulence models (SA, $k-\epsilon$, SST and RSM), and compared with experimental measurements of the mean flow quantities and the wall shear stress. The pressure gradients were imposed on high Reynolds number, 2-D turbulent boundary layer developing on a flat plate by changing the ceiling geometry. Two converging humps (at $x = 2m$ and $x = 5.5m$ from the leading edge of the plate) and two diverging humps at the same locations were considered. The SST model produced the best agreement with experiments. A complimentary study about how the models deal with numerical transition was done by solving a zero pressure gradient (ZPG) configuration. We find that the major differences between the results from the models when predicting mean flow quantities are essentially produced by the numerical transition process. This process does not belong to the models themselves, and it is a procedure by which the software transforms the simple laminar boundary conditions at the inlet into inflow conditions which characterize the turbulent flow when turbulence has already been developed. Therefore, models requiring the simplest inflow conditions lead to better results and consequently models such as the RSM suffer the most and ultimately lead to inferior results.

12:06PM PA.00003 Two-point correlations of adverse pressure gradient turbulent boundary layer at high Reynolds number[1] , MURAT TUTKUN, Norwegian Defence Research Establishment (FFI), WILLIAM K. GEORGE, MICHEL STANISLAS, Laboratoire de Mecanique de Lille, CNRS — Two-point correlations are analyzed to investigate structure of adverse pressure gradient turbulent boundary layer at high Reynolds number. The experiment was carried out in the large wind tunnel of Laboratoire de Mécanique de Lille (LML) using synchronized PIV systems and a hot-wire rake of 143 single probes. A specially designed, 30 cm thick, bump was used to obtain the decelerating flow within the test section of the wind tunnel. The thickness of the boundary layer was about 30 cm and Reynolds number based on momentum thickness, Re_θ, was 30 000 for 10 m/s external free stream velocity. Simultaneously measured hot-wire data show that extent of the two-point correlations on streamwise – spanwise plane in this flow is similar to that computed for flat plate turbulent boundary layers case(J. Turbulence, Vol 10, No 21, pp 1-23, 2009). Two-point correlations are also studied on streamwise – wall-normal plane. Shape of the correlations in this plane, especially in the outer layer, is found to be different than those for flat plate turbulent boundary layer.

[1]This work has been performed under the WALLTURB project funded by the CEC under the 6th F.P. (Contract No:AST4-CT-2005-516008).

12:19PM PA.00004 Structures of turbulent boundary layers with adverse pressure gradients[1] , JIN LEE, HYUNG JIN SUNG, KAIST — Turbulent structures in spanwise/wall-normal plane of the turbulent boundary layers (TBLs) subjected to adverse pressure gradients (APGs) were investigated by analyzing the DNS database of Lee & Sung (2009). Probability density functions of the strength of the vortex cores normalized by their r.m.s. values displayed that strong swirling motions are frequently observed on the APG TBLs than zero pressure gradient TBLs. Influence of APGs on the population trends of spanwise vortex cores showed that those have a local maximum at the outer region for APG TBLs which might be due to the maximum Reynolds shear stress. Moreover, two-point correlations and linear stochastic estimations were scrutinized to provide statistical evidence for hairpin packet motions in the vertical plane of the TBLs with APGs. We found that wall-normal extent of the contours is elongated vertically owing to the strong swirling motion of the individual vortex located in the wake region or wake-type detached structures.

[1]This work was partially supported by the KISTI under the Grand Challenge Supercomputing Program.

12:32PM PA.00005 Experimental investigation of turbulent sink flow boundary layers[1] , PRANAV JOSHI, JOSEPH KATZ, Johns Hopkins University — Our objective is to study near wall structures in turbulent boundary layers on smooth walls under favorable pressure gradient. A sink flow is generated in a refractive index matched facility by means of an inclined upper wall. 2D-PIV measurements have been performed, initially at a coarse resolution of 320 μm, to characterize the mean flow and Reynolds stresses, for a constant acceleration parameter, $K=\nu dU/dx/U^2$, of 0.5 X10^{-6}. Re_θ drops from 5300 to 2600 over the 260 mm long accelerating range. Consistent with prior studies, acceleration thins the boundary layer. Trends of Reynolds stress vary in the literature. In our data, acceleration increases the magnitude of Reynolds stresses in outer parts of the boundary layer and decreases it in inner parts. An upsurge in all stresses occurs below 200 wall units in later stages of acceleration. When non-dimensionalized with local free-stream or friction velocity, the Reynolds stresses drop everywhere, except for an increase in <v'v'> and -<u'v'> below 200 wall units. We are following with higher resolution near wall measurements, and plan to acquire 3D data using holography in the near future.

[1]Sponsored by NSF

12:45PM PA.00006 Interaction between a rough boundary layer and multiple cylinders wakes: application to ecological restoration[1] , ALINE COTEL, University of Michigan, OLIVIER EIFF, FREDERIC MOULIN, Institut de Mécanique des Fluides de Toulouse — 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" have more effective control systems and greater ability to generate propulsive power to maneuver. Understanding fish responses and interactions with turbulence is an important biological issue pertinent to evolution of swimming mechanisms and capabilities, and ecological roles and distributions of fishes. There is a current lack of quantitative evaluation of such systems. In most natural systems, sediments and various factors in streambed topography create a rough turbulent boundary layer along the bottom. This work used complimentary laboratory experimental studies and previous field observations (Cotel et al. 2005) to determine how a rough turbulent boundary layer interacts with flow structures created by obstacles in a channel using PIV. Preliminary analysis shows a strong interaction between the turbulent boundary layer created by roughness elements and the wakes behind cylinder arrays.

[1]Supported by NSF IREE.

12:58PM PA.00007 Experimental study of the horizontally averaged flow structure in a model wind-turbine array boundary layer[1] , RAUL BAYOAN CAL, Portland State University, JOSE LEBRON-BOSQUES, Rensselaer Polytechnic Institute, HYUNG-SUK KANG, Johns Hopkins University, LUCIANO CASTILLO, Rensselaer Polytechnic Institute, CHARLES MENEVEAU, Johns Hopkins University — Wind-tunnel measurements are performed in order to quantify the vertical transport of momentum and kinetic energy across a boundary layer that includes a three-by-three array of model wind turbines. The data are obtained using stereo-PIV, on 18 planes surrounding a wind turbine. The data are used to compute mean velocity and turbulence properties averaged on horizontal planes. We compare the effects of turbulence stresses with those arising from the averaging of spatially varying mean flow distributions ("canopy stresses"). Results are compared with simple momentum theory and with models for effective roughness length scales that are often used to model wind turbine arrays in computer models for the large scales of the atmosphere. The impact of vertical transport of kinetic energy due to turbulence and mean flow correlations is quantified. It is found that the fluxes of kinetic energy associated with the Reynolds shear stresses are of the same order of magnitude as the power extracted by the wind turbines, highlighting the importance of vertical transport.

[1]Supported by the NSF (CBET-0730922).

1:11PM PA.00008 Turbulent properties of a wind-turbine wake developed in a boundary layer flow , LEONARDO CHAMORRO, FERNANDO PORTE-AGEL, University of Minnesota — Wind turbine wake characteristics are expected to depend on the incoming atmospheric boundary layer flow statistics (e.g., mean velocity distribution, turbulent stresses and turbulent fluxes). Atmospheric stability is also expected to affect the structure of a turbine wake. In this study, results are presented from wind tunnel experiments carried out at the St. Anthony Falls Laboratory atmospheric boundary layer wind tunnel using a model wind turbine placed inside the boundary layer developed over a smooth and rough surface. The structure and behavior of the wind turbine wake are studied also under different conditions of thermal stratification. Thermal stratification levels in the boundary layer were achieved by controlling the temperature of both the tunnel floor and the air flow. A triple-wire (x-wire and cold wire) anemometer and Particle Image Velocimetry (PIV) were used to characterize the turbulent wake downwind of the turbine at different locations. This study provides valuable information about the spatial characteristics of the structure of wind-turbine wakes. This information is being used to test and guide the development of improved parameterizations of wind turbines in high-resolution numerical models, such as large-eddy simulations (LES).

1:24PM PA.00009 Statistical and quadrant-hole analysis of the turbulence characteristics in a 3 x 3 wind turbine array boundary layer[1] , MAX GIBSON, Portland State University (PSU), HYUNG-SUK KANG, Johns Hopkins University (JHU), CHARLES MENEVEAU, JHU, RAUL BAYOAN CAL, PSU — Data from a wind-tunnel experiment on the flow within a 3 x 3 array of lightly loaded wind turbine models operating inside a turbulent boundary layer over a rough surface are analyzed. The data are acquired using X hot-wire anemometry and the focus of the analysis is on the possible differences of the flow structures above and below the canopy of wind turbines. Here this question is addressed using quadrant analysis. Conditional averages of turbulent dissipation (a 1-D surrogate) at various heights at 5 diameters downstream is performed for each of the 4 quadrants as well as different "hole-sizes." The results imply significantly less inter-scale correlations in the low-shear region at the bottom of the wind turbine wake than at other wake locations. Inter-scale correlations above and below the wake are also significantly greater than at that low-shear region. Spectral analysis is performed to determine which scales are mostly responsible for the various levels of Reynolds stresses as functions of position in the wind turbine wake.

[1]Supported in part by the NSF (CBET-0730922).

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PB Turbulence Modelling IV 101B

11:40AM PB.00001 Chasing eddies and their wall signature in turbulent boundary layers at Mach 3 through 14[1] , I.B. BEEKMAN, S. PRIEBE, M.P. MARTIN, Princeton University, Princeton, NJ 08544 USA — We use a direct numerical simulation database of turbulent boundary layers,[2,3,4] statistical tools,[5] scientifically-rooted packet-pattern recognition,[6] and validated visualization algorithms[7] to identify hairpin packets and their wall signature. We investigate the variation of time scales and length scales associated with coherent structures and the role of hairpin packets on the generation of skin friction, wall-pressure loading and heat transfer.

[1]Funded by AFOSR Grant # FA9550-06-1-0323 and NASA Grant # NNX08AD04A.
[2]Martin, M.P., JFM, vol. 570, pp. 347-364, 2006
[3]Martin, M.P., AIAA Paper 2004-2337
[4]Beekman & Martin, APS DFD08
[5]Brown & Thomas, Phys. Fluids, vol. 20, pp 243-251, 1977
[6]Ringuette, Wu & Martin, JFM, vol. 594, pp. 59-69, 2008
[7]O'Farrell, C. Senior Thesis, Princeton University 2008

11:53AM PB.00002 ABSTRACT WITHDRAWN –

12:06PM PB.00003 A Correlation Matrix Approach to Esimtating Velocity Fields Using Sensor Measurements , DIETMAR REMPFER, PARITOSH MOKHASI, IIT, Chicago — A new approach to estimating velocity fields from sensor measurements is proposed based on approximating the correlation of the unknown velocity fields with a sample ensemble of snapshots. Proper orthogonal decomposition (POD) is performed on the ensemble to extract the spatial eigenfunctions. If the POD coefficients are known at a certain instance in time, then the velocity field at that time can be estimated. The POD coefficients can be computed if the correlations between the ensemble and the velocity field are known. However, since the velocity field is not known, an extension of POD called "episodic-POD" is used to produce models that enable one to approximate the correlation between the unknown velocity fields at the ensemble based on sensor measurements taken from the domain. The episodic-POD analysis on the correlation matrix reveals that the underlying structure of the velocity correlation is low-dimensional even if the original flow is high-dimensional. Furthermore, the structures are seen to be similar for different problems indicating some form of universality. It is shown that the method is robust in the presence of noisy measurements and sparse temporal data. It is also proposed that this new approach suggests that solving a high-dimensional system could be replaced with solving a single low-dimensional system with multiple initial conditions.

12:19PM PB.00004 Assessing Turbulent Convective Heat Transfer Effectiveness with POD-based low order models , MARKUS SCHWAENEN, ANDREW DUGGLEBY, Texas A&M — The efficiency of convective turbulent heat transfer processes is significantly affected by turbulent flow structures. These either increase drag within the flow for the gain of higher heat transfer or need to be controlled in order to maintain a given surface temperature level for cooling applications. Proper Orthogonal Decomposition (POD) has proven to be a useful tool for the analysis of structures in turbulent flows. In the approach developed, we extend the use of POD by including the fluid temperature, thus making a quantitative assessment of the interaction between flow structures and surface heat transfer possible. Specifically, we use the low order flow description provided by POD to create a framework within one can identify the most important spatial or temporal flow structures that have undesired implications on the convective energy transport at hand. We achieve this by calculating enthalpy thicknesses based on fluctuating quantities from the most energetic mode pairs. The method has been applied to pin fin heat transfer which might exist, for example, in an internal gas turbine cooling passage.

12:32PM PB.00005 Spatial Variable Thresholding for SCALES[1] , ALIREZA NEJADMALAYERI, OLEG V. VASILYEV, ALEXEI VEZOLAINEN, University of Colorado at Boulder, GIULIANO DE STEFANO, Seconda Universita Napoli, Italy — The Stochastic Coherent Adaptive Large Eddy Simulation (SCALES) is a novel wavelet-based approach that resolves energy containing turbulent motions using wavelet multiresolution decomposition and self-adaptivity. The extraction of the most energetic structures is achieved using wavelet thresholding filter with a priori prescribed threshold level. This strategy, although successful, has a major drawback: the thresholding criterion is global and does not fully utilize the spatial/temporal intermittency of the turbulent flow. In the current numerical effort, for the first time (to the best of our knowledge), the concept of physics-based spatially variable thresholding in the context of wavelet-based numerical techniques for solving PDEs is introduced. The procedure consists of tracking the wavelet thresholding-factor within a Lagrangian frame by exploiting a Lagrangian Path-Line Diffusive Averaging approach that uses linear averaging along characteristics. The results for incompressible flow around NACA 0015 airfoil show a very robust and fast methodology for adjusting the thresholding-factor based on dynamically important flow characteristics, for instance, the magnitude of vorticity or strain rate.

[1]This work was supported by NSF under grant No. CBET-0756046.

12:45PM PB.00006 Coherent vortex simulation of 3D homogeneous isotropic turbulence , NAOYA OKAMOTO, KATSUNORI YOSHIMATSU, Department of Computational Science and Engineering, Nagoya University, Nagoya, Japan, KAI SCHNEIDER, M2P2-CNRS & CMI, Aix-Marseille University, Marseille, France, MARIE FARGE, LMD-IPSL-CNRS, Ecole Normale Superieure, Paris, France, YUKIO KANEDA, Department of Computational Science and Engineering, Nagoya University, Nagoya, Japan — Coherent vortex simulation based on the wavelet filtered Navier-Stokes equations are presented for three-dimensional decaying isotropic turbulence. The vorticity field is decomposed into coherent vortices and an incoherent background noise using an orthogonal wavelet representation. The time evolution of the coherent vortices is then integrated deterministically, while discarding the incoherent flow contributions at each time step is shown to be sufficient to model turbulent dissipation. The wavelet filter dynamically adapts to the flow evolution and thus changes with time. A safety zone is required to track the vortices and the small scales produced by their nonlinear interaction. Different strategies for choosing the safety zone are tested and their influence on the precision and efficiency is assessed. We show that an adequate choice allows to reduce the number of degrees of freedom by a factor 6 with respect to direct numerical simulation, while preserving the high order statistics of the flow.

12:58PM PB.00007 Higher-order system analysis , WOUTIJN BAARS[1], CHARLES TINNEY[2], The University of Texas at Austin — A higher-order system identification technique will be presented in the context of stochastic estimation, as it is quite useful in the field of (experimental) fluid dynamics. This higher-order spectral stochastic estimation technique was originally developed in the context of Systems Identification. It is shown how this technique defaults to spectral Linear Stochastic Estimation when only the linear kernels are computed. In case of higher-order computations, the system is constructed using a frequency-domain Volterra series and is expressed as a linear algebraic system of equations that are solved for the linear and higher-order transfer kernel coefficients. In the trade-off to seek for higher-order transfer kernels, the increased complexity restricts the analysis to single input/single output. POD based methods can be inserted to alleviate this void whereby time-varying POD coefficients of the output are estimated from POD coefficients of the input. Strengths and weaknesses of the current implementation of this technique will be discussed using simulated data and an application of this method to an axisymmetric jet flow to identify coherent turbulent structures.

[1]PhD student
[2]Assistant Professor

1:11PM PB.00008 Representing broad band effects by low order Galerkin models of fluid flows[1] , GILEAD TADMOR, Northeastern University, BERND R. NOACK, MICHAEL SCHLEGEL, OLIVER LEHMANN, Berlin Institute of Technology, SCOTT KELLEY, Northeastern University, MAREK MARZYNSKI, Poznan University of Technology — We discuss a system reduction strategy for spectral and Galerkin models of incompressible fluid flows. This approach leads to dynamic models of lower order, based on a partition in slow, dominant and fast modes. In the reduced order models,slow dynamics are incorporated as nonlinear manifold consistent with mean-field theory. Fast dynamics are stochastically treated and can be lumped in eddy viscosity approaches. The employed interaction models between slow, dominant and fast dynamics 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 proposed system reduction strategy is employed to the cylinder wake benchmark.

[1]Supported at Northeastern University by NSF grant 0524070 and the US AFOSR grant FA9550-05-1-0399, and at the Berlin Inst. of Technology, by the Collaborative Research Centre (Sfb 557) "Control of complex turbulent shear flows" and by DFG grants N.

1:24PM PB.00009 Cycles and unstable manifolds in bursting Couette flow[1] , LENNAERT VAN VEEN, University of Ontario Institute of Technology, GENTA KAWAHARA, ATSUSHI MATSUMURA, Osaka University — Since the publication of a landmark paper by Kawahara and Kida on the relevance of unstable periodic solutions to shear flow in 2001, the scale of dynamical systems-type computations in turbulence research has increased spectacularly. Equilibrium and periodic solutions have been computed in great spatial detail for Couette flow, pipe flow and many other geometries. One of the main goals of these computations is to explain the process of turbulent bursting in shear flows. Often, this transition occurs in the presence of a asymptotically stable laminar flow, so ordinary bifurcation scenarios cannot explain them. Instead, the current focus is on so-called "edge states," i.e. saddle-type equilibria or periodic solutions that appear to live on a boundary between turbulent and laminar behaviour in phase space. In principle, we should be able to clarify the bursting process if we know the geometry of the (un)stable manifolds of such states. However, the systematic computation of these manifolds is a hard task. We will present a recently developed algorithm for the computation of unstable manifolds and its adaption to shear flow, both in a toy model and in a full-scale simulation of turbulent Couette flow.

[1]Supported by the Fonds Quebecois de la Recherche sur la Nature et les Technologies.

1:37PM PB.00010 A-Posteriori Study of the Sensitivity Equation Method and Complex-Step Differentiation in Capturing Coherent Structures in the Sensitivity Field of a Planar Mixing Layer , MOHSEN ZAYERNOURI, MEREDITH METZGER, University of Utah — The performance of two techniques in computational sensitivity analysis, the sensitivity equation method (SEM) and complex-step differentiation (CSD), has been evaluated for the incompressible, two-dimensional, temporal mixing layer. This a-posteriori study aims to discover how well these two approaches capture the coherent structures in the sensitivity field, where the quantity of interest is the sensitivity of vorticity to changes in the Reynolds number, Re_{δ_0}. In SEM, the governing equations are first differentiated with respect to the parameter of interest, in this case Re_{δ_0}, then discretized and solved numerically to obtain the sensitivity coefficients. In CSD, the governing equations are treated as complex; and, the sensitivity coefficients are estimated by dividing the imaginary part of the velocity field by a small perturbation in the parameter of interest. In this manner, CSD avoids subtractive cancellation errors associated with finite difference approximations. Simulations were run at a baseline test case of $Re_{\delta_0} = 200$ using an unsteady finite-volume-based fractional step algorithm. The results show that CSD has many advantages over SEM including ease of implementation, faster performance, and higher accuracy at a same resolution.

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PC Turbulence Simulations VII 101C

11:40AM PC.00001 Numerical studies of the flow past a rotating sphere at $Re = 500$, ERIC POON, University of Melbourne, GIANLUCA IACCARINO, Stanford University, ANDREW OOI, University of Melbourne, MATTEO GIACOBELLO, Defence Science and Technology Organisation — The flow past a rotating sphere is of interest in many engineering applications. A number of numerical studies have been performed to investigate the flow past a streamwise and transversely rotating sphere in the laminar flow regime ($Re \leq 300$, where Re is the Reynolds number based on freestream velocity, U_∞, and sphere diameter, d). In order to extend the understanding of a sphere's trajectory due to the rotating motion of the sphere, the flow past a rotating sphere is investigated for $Re = 500$. This Reynolds number sits in the unsteady asymmetric regime for a stationary sphere. The non-dimensional rotation rates, $0 \leq \Omega^* \leq 1$, are considered. The sphere rotation axis is orientated in two of the principle axis directions, namely streamwise and transverse directions. The effect of the rotation axis and Ω^* on the flow structures and the aerodynamic forces of a rotating sphere will be presented.

11:53AM PC.00002 Direct numerical simulation of turbulent flow over a backward-facing step[1] , MICHAL A. KOPERA, CHRISTOPHER CANTWELL, ROBERT M. KERR, DWIGHT BARKLEY, University of Warwick, HUGH BLACKBURN, Monash University — Turbulent flow in a channel with a sudden expansion(backward-facing step) is studied by direct numerical simulation (DNS) of incompressible Navier-Stokes equations. Initial results are presented for a 3D DNS of a backward-facing step flow with Reynolds number 6000, based on average bulk upstream velocity and step height. The expansion ratio is 2. Turbulent inflow is provided by regeneration of velocity and pressure fields from a plane downstream from the inflow. Simulations are made using the Semtex DNS spectral element solver. The goal is to generate hi-resolution DNS data of a high Reynolds number flow over a backward-facing step for LES comparisons.

[1]Acknowledge support of EPSRC/UK

12:06PM PC.00003 Numerical Simulations of Turbulent Rayleigh-Benard Convection , JANET SCHEEL, Occidental College — The results from numerical simulations of three-dimensional, fully turbulent Rayleigh-Benard convection will be presented. These results are compared to experiments and theoretical stochastic models, particularly the scaling of heat transport and thermal/viscous boundary layers with Rayleigh number. The orientation and strength of the large-scale circulation has also been measured. The orientation of this large-scale circulation exhibits diffusive meandering in addition to chaotic, abrupt switches.

12:19PM PC.00004 Direct Numerical Simulation of Spiral Turbulence , S. DONG, X. ZHENG, Purdue University — Spiral turbulence in the Taylor-Couette setting is one of the most fascinating phenomena of fluid dynamics, where intertwined helical turbulent and laminar patterns propagate between counter-rotating concentric cylinders. It is characterized by spatiotemporal intermittency and the co-existence of turbulent and laminar domains in space and time. We report a direct numerical simulation of spiral turbulence in the Taylor-Couette geometry over a range of Reynolds numbers at which well-defined turbulent/laminar spiral patterns are observed. The statistical features of turbulent spirals are characterized with a conditional averaging technique.

12:32PM PC.00005 Direct Numerical Simulation of Turbulent Pipe Flows subjected to Transverse Oscillations , MARK CZAJKOWSKI, OLIVIER DESJARDINS, University of Colorado at Boulder — Fundamental effects of transverse oscillations on a turbulent pipe are being studied using direct numerical simulations (DNS). Previous studes of pipes subjected to oscillations around their central axis have shown a reduction in pressure drop. This study is a generalization of previous work to the case of an oscillation around an axis parallel to but not coinciding with the pipe centerline.The role of oscillation frequency, amplitude, and radius on the statistics of turbulent pipe flows, as well as bulk properties like pressure drop are investigated. Two key non-dimensional numbers are identified. The first relates the oscillation amplitude to a turbulent length scale (Taylor micro-scale); and the second number, a Strouhal number, compares the oscillation frequency to a turbulent time scale. These non-dimensional numbers, along with oscillation radius, are varied over a range of values, and DNS were performed using the arbitrarily high order accurate code NGA [Desjardins, et al., JCP 2008]. The resulting turbulent flows were analysed using a variety of turbulent statistics and compared to stationary and pipes rotating around their central axis.

12:45PM PC.00006 Numerical Simulation of Flow Over a Savonius Wind Turbine Using a Spectral Element Method , SRIHARSHA KANDALA, DIETMAR REMPFER, IIT, Chicago — A parallel spectral element code, SpecSolve, is developed with the objective of modeling flows in complex geometries. This code supports both structured and unstructured meshes and allows exact representation of boundary surfaces which are particularly useful for modeling turbo machinery flows. In this talk we present the results from 2D Navier-Stokes simulations of flow over a Savonius turbine. The simulation uses a rotating mesh in regions surrounding the blade and a stationary mesh away from the rotor. Results of a 2D Optimization study involving overlap ratio and the number of blades are also presented. These results are compared with experimental data.

12:58PM PC.00007 Numerical Investigation of Base Drag for an Axisymmetric Underwater Vehicle with Bluff Afterbody , MIN-JAE KIM, KURNCHUL LEE, Agency for Defense Development(ADD) — The objective of this study is to predict the drag of an axisymmetric underwater vehicle with bluff afterbody. FLUENT, commercial CFD code, is used to simulate high Reynolds number turbulent flows around the vehicle. The computed drag coefficients are compared to available experimental data at various Reynolds numbers. Four widely used two-equation turbulence models are investigated to evaluate their performance of predicting the anisotropic turbulence in a recirculating flow region, which is caused by flow separation arising from the base of the vehicle. The simulations with Realizable k-ε and k-ω SST turbulence models predict the anisotropic turbulent flows comparatively well and the drag prediction results with those models show good agreements with the experimental data.

1:11PM PC.00008 Aerodynamics of a Cryogenic Semi-Tanker , JASON ORTEGA, KAMBIZ SALARI, Lawrence Livermore National Laboratory — The design of a modern cryogenic semi-tanker is based primarily upon functionality with little consideration given to aerodynamic drag. As a result, these tankers have maintained the appearance of a wheeled cylinder for several decades. To reduce the fuel usage of these vehicles, this study investigates their aerodynamics. A detailed understanding of the flow field about the vehicle and its influence on aerodynamic drag is obtained by performing Reynolds-Averaged Navier-Stokes simulations of a full-scale tractor and cryogenic tanker-trailer operating at highway speed within a crosswind. The tanker-trailer has a length to diameter ratio of 6.3. The Reynolds number, based upon the tanker diameter, is 4.0×10^6, while the effective vehicle yaw angle is 6.1°. The flow field about the vehicle is characterized by large flow separation regions at the tanker underbody and base. In addition, the relatively large gap between the tractor and the tanker-trailer allows the free-stream flow to be entrained into the tractor-tanker gap. By mitigating these drag-producing phenomena through the use of simple geometry modifications, it may be possible to reduce the aerodynamic drag of cryogenic semi-tankers and, thereby, improve their fuel economy. This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.

1:24PM PC.00009 Modeling the Free Surface Penetration of Cylinders and Spheres , SAMUEL KOSKI, Naval Surface Warfare Center Dahlgren & Virginia Tech, PAVLOS VLACHOS, Virginia Tech — Free surface penetration is a multiphase flow problem whereby a solid body is translated through a gas penetrating a liquid. The nature of the ensuing fluid dynamics poses a highly complex analytical modeling effort. It is known that for a given surface roughness, body geometry, and velocity, a cavity can be formed aft of the body and maintained for a given distance with traveling from air into water. An analytical model [Dynamics of Transient Cavities, V. Duclaux et al.] of the classical free surface penetration of spheres and cylinders and computational simulations using ANSYS CFX version 12 are preformed and presented. Comparisons of the pinching (closure) and cavity radius as a function of time are considered and the effect of surface roughness is explored.

1:37PM PC.00010 Simulation of Acid-Base Reactions in a Microscale Confined Impinging Jet Reactor , JEN OLSEN, LUCAS GRIFFITH, MICHAEL OLSEN, Iowa State University, RODNEY FOX, Iowa State Universtiy — Numerical simulations were performed of reactive mixing in a microscale confined impinging jets reactor (CIJR). A simple acid-base reaction was modeled and solved for a laminar flow case. Local pH throughout the reactor was determined as a function of hydrogen and hydroxide mixture fraction, and this relationship was used to generate a pH field that could be compared with experimental results. An experimental visualization of the acid-base reaction was performed in the CIJR with phenolphthalein used as a pH indicator. Visualization images of the actual flow were then compared with the predictions of the simulation. Good agreement was observed between the experiment and the simulation. Future work will include adding a turbulence model to the simulation.

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PE Biofluids X: General IV - Biomedical Flows 101E

11:40AM PE.00001 Scaling analysis of magnetic fluid hyperthermia , MONRUDEE LIANGRUKSA, Department of Engineering Science and Mechanics, Virginia Polytechnic Institute and State University, RANJAN GANGULY, Department of Power Engineering, Jadavpur University, Kolkata, India, ISHWAR K. PURI, Department of Engineering Science and Mechanics, Virginia Polytechnic Institute and State University — Magnetic fluids have been investigated for hyperthermia and thermoablation applications. These have advantages over traditional treatments in terms of localization of the therapy and less serious side effects. Herein, an analysis of bioheat transfer during magnetic fluid hyperthermia is performed. The results provide insight into the roles of dimensionless numbers in the bioheat equation and can lead to parameter optimization for in vitro and in vivo magnetic fluid hyperthermia experiments. In addition, a clinical efficacy index and a damage is provided for magnetic heating.

11:53AM PE.00002 Effects of encapsulation elasticity on the stability of an encapsulated contrast microbubble for medical imaging and drug delivery , AMIT KATIYAR, KAUSIK SARKAR, University of Delaware — Encapsulated microbubbles for ultrasound imaging show a shelf life of months while free bubbles, in aqueous medium, last for milliseconds. For microbubbles, with inelastic encapsulation, lifetime of hours is possible only at extremely low surface tension ($<$1 mN/m) or at extreme oversaturation. However, microbubbles with elastic encapsulation can resist dissolution. Analytical expressions involving saturation level, surface tension and interfacial dilatational elasticity are determined for attaining non-zero equilibrium radius for these microbubbles. All encapsulated bubbles dissolve in undersaturated medium. In a saturated medium, an encapsulated bubble is found to achieve a long-time stable radius when interfacial dilatational elasticity is larger than the equilibrium surface tension. For bubbles with interfacial dilatational elasticity smaller than the equilibrium surface tension, stable bubble of non-zero radius can be achieved only when the saturation level is greater than a critical value. Even if they initially contain a gas other than air, bubbles that reach a stable radius finally become air bubbles. The model is applied to an octafluoropropane filled lipid-coated 2.5 lm bubble. Effects of elasticity, shell permeability, initial mole fraction, initial radius and saturation level are discussed.

12:06PM PE.00003 Analysis of Alternative Polling Strategies for Derivative-Free Optimization of the Fontan Surgery , WEIGUANG YANG, UCSD, JEFFREY FEINSTEIN, Stanford University, ALISON MARSDEN, UCSD — We have recently proposed a new design for the Fontan surgery, performed to treat children with severe heart defects. The new design replaces the traditional straight graft between the inferior vena cava and the pulmonary arteries, with a Y-shaped graft. In preliminary work, this design offered superior hemodynamic performance compared to designs used in current practice. Here, we optimize an idealized Y-graft design under a range of rest to exercise pulsatile flow conditions using the surrogate management framework (SMF) and mesh adaptive direct search (MADS). Constraints are added to the problem using a filter method. In particular, we assess and compare the performance of two recently developed polling strategies used in the SMF algorithm: LTMADS and OrthoMADS. Although LTMADS works well in many applications, it chooses the polling directions randomly and may result in large angles between poll directions. OrthoMADS generates deterministic and orthogonal polling directions, which avoids the drawbacks above while still producing a dense set of directions. Finally, we extend the SMF method to incorporate multiple objectives based on clinical outcome data for Fontan patients.

12:19PM PE.00004 Optimal shape design for cardiovascular surgery applications in the presence of uncertainties: a stochastic derivative-free approach , SETHURAMAN SANKARAN, UCSD, JEFFREY FEINSTEIN, Stanford University, ALISON MARSDEN, UCSD — In the field of cardiovascular medicine, predictive finite element simulations that compute the hemodynamics of blood flow, particle residence times, as well as shear stresses induced on arterial walls could aid in surgical intervention. These simulations lack accurate input data and are often polluted with uncertainties in model geometry, blood inlet velocities and outlet boundary conditions. We develop a robust design framework to optimize geometrical parameters in cardiovascular simulations that accounts for diverse sources of uncertainties. Stochastic cost functions are incorporated into the design framework using their lower order statistical moments. The adaptive stochastic collocation technique embedded within a derivative-free optimization technique is employed. Numerical examples representative of cardiovascular geometries, including robust design on various anastomoses is presented and the efficiency of the adaptive collocation algorithm is shown.

12:32PM PE.00005 Image-Based Flow Modeling , SETH DILLARD, JOHN MOUSEL, JAMES BUCHHOLZ, H.S. UDAYKUMAR, University of Iowa — A preliminary method has been developed to model complex moving boundaries interacting with fluids in two dimensions using video files. Image segmentation techniques are employed to generate sharp object interfaces which are cast as level sets embedded in a Cartesian flow domain. In this way, boundary evolution is effected directly through imagery rather than by way of functional approximation. Videos of an American eel swimming in a water tunnel apparatus and a guinea pig duodenum undergoing peristaltic contractions *in vitro* serve as external and internal flow examples, which are evaluated for wake structure and mixing efficacy, respectively.

12:45PM PE.00006 Flow visualization of sterile air flows in surgical environments[1] , JAMES MCNEILL, JEAN HERTZBERG, ZHIQIANG ZHAI, University of Colorado, Boulder — The current design of surgical environments uses laminar flow air diffusers, originally intended for clean room environments, to provide clean, low-turbulence ventilation air across the patient surgical wound and aseptic regions of the surgical staff. The objective of the current design is to minimize turbulence which increases mixing between the sterile air field and any non-sterile areas. Full-scale laboratory experiments using laser sheet illumination were conducted to investigate the interface between the sterile air field underneath the laminar flow diffusers and the non-sterile room air in the area surrounding the diffuser array. The shear layer between the sterile air and the room air resulted in turbulent mixing. In addition, the shear layer boundary angled towards the patient causing the migration of contaminants towards the aseptic region. Further research is being conducted to understand the impact of the free shear layer along the boundary of the laminar flow diffuser array on contaminant transport into the sterile region.

[1]This work is supported by ASHRAE.

12:58PM PE.00007 Particle Size, Magnetic Field, and Blood Velocity Effects on Particle Retention in Magnetic Drug Targeting , ERICA CHERRY, PETER MAXIM, JOHN EATON, Stanford University — Magnetic drug targeting (MDT) is a promising cancer treatment technique in which magnetic drug particles are steered through the blood stream or held near a tumor site using external magnetic fields. A physics-based model of a general MDT system was developed with the goal of realizing the practical limitations of MDT when electromagnets are the source of the magnetic field. The simulation tracks magnetic particles subject to gravity, drag force, magnetic force, and hydrodynamic lift in specified flow fields and external magnetic field distributions. A model problem was analyzed to determine the effect of drug particle size, blood flow velocity, and magnetic field gradient strength on efficiency in holding particles stationary in a laminar Poiseuille flow modeling blood flow in a medium-sized artery. It was found that particle retention rate increased with increasing particle diameter and magnetic field gradient strength and decreased with increasing bulk flow velocity. The results suggest that MDT systems with electromagnets are unsuitable for use in small arteries because it is difficult to control particles smaller than about 20 microns in diameter.

1:11PM PE.00008 Bubble Evolution During Acoustic Droplet Vaporization , ADNAN QAMAR, JOSEPH BULL, The University of Michigan — A first theoretical model of bubble evolution in Acoustic Droplet Vaporization (ADV) inside a circular microchannel is presented. This work is motivated by a novel gas embolotherapy technique, which is intended to treat cancers by occluding blood flow using gas bubbles. The intended therapy involves the injection of superheated Dodecafluoropentane (DDFP, C_5F_{12}, boiling point 29°C) droplets, each encapsulated in an albumin shell, into the blood stream. The blood circulation carries these droplets into the tumor region where high-intensity ultrasound is used to trigger ADV to form bubbles near the desired occlusion sites. The proposed model describes the rapid phase transition from highly superheated DDFP droplet to the vapor phase via a homogeneous nucleation within the DDFP droplet. For every time step the radial component of the Navier-Stokes equation is integrated from the nucleated bubble surface to the expanding boundary of the droplet with proper boundary conditions taking into account for the vaporization process. Further from the droplet boundary to the end of microchannel a modified unsteady Bernoulli equation with the head loss term is utilized. Close agreement with experimental data for all the acoustic parameters and different initial droplet sizes is obtained. The proposed model is expected to elucidate the role of different parameters involved in the complex ADV process. This work is supported by NIH grant R01EB006476.

1:24PM PE.00009 Numerical Simulation of Cellular Blood Flow through a Rigid Artery , DANIEL REASOR, JONATHAN CLAUSEN, CYRUS AIDUN, Georgia Institute of Technology — In blood flow, red blood cells (RBCs), the most numerous constituent of blood, influence continuum-level measures by altering the suspension at microscopic scales. The presence of RBCs alters the stress and diffusion individual cells experience, which can influence cardiovascular diseases by affecting other cells present in blood like platelets and white blood cells. Simulations of blood at a cellular level provide a tool that allows exploration of both the rheology and the stress and diffusion of individual suspended cells. In this work, a hybrid lattice-Boltzmann/finite element method is used to simulate suspension flows characteristic of blood with deformable RBCs at realistic hematocrit values. We have shown the ability to simulate thousands deformable suspensions capturing non-Newtonian flow characteristics such as shear thinning, and the results agree well with experimental observations. Simulations through rigid arteries have been deformed with as many as 2500 RBCs. This work outlines results obtained for pressure-gradient driven blood flow through a rigid artery with 20%, 30%, 40%, and 50% hematocrit values. Results include the effect these deformable RBCs have on mean velocity, flow rate, radial variation of RBC concentration, and the effective viscosity for simulations at moderate to low cell capillary numbers, $Ca \leq 0.08$.

1:37PM PE.00010 Computational analysis of an aortic valve jet , SHAWN C. SHADDEN, Illinois Institute of Technology, MATTEO ASTORINO, JEAN-FRÉDÉRIC GERBEAU, INRIA Rocquencourt — In this work we employ a coupled FSI scheme using an immersed boundary method to simulate flow through a realistic deformable, 3D aortic valve model. This data was used to compute Lagrangian coherent structures, which revealed flow separation from the valve leaflets during systole, and correspondingly, the boundary between the jet of ejected fluid and the regions of separated, recirculating flow. Advantages of computing LCS in multi-dimensional FSI models of the aortic valve are twofold. For one, the quality and effectiveness of existing clinical indices used to measure aortic jet size can be tested by taking advantage of the accurate measure of the jet area derived from LCS. Secondly, as an ultimate goal, a reliable computational framework for the assessment of the aortic valve stenosis could be developed.

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PF Microfluidics: Slip and Experimental Methods 101F

11:40AM PF.00001 The effect of tracer properties on interfacial particle-image velocimetry results in microflows[1] , YUTAKA KAZOE[2], MINAMI YODA, Georgia Institute of Technology — Interfacial effects are important in many cases for microscale transport. One of the few experimental techniques that can resolve interfacial transport at these spatial scales is multilayer nano-particle-image velocimetry (MnPIV), which exploits the exponentially decaying intensity of evanescent-wave illumination to obtain velocities at different distances from the fluid-solid interface within 400 nm of the wall. Although MnPIV results have been validated in steady and creeping Poiseuille flow, the results of this technique, like all tracer-based velocimetry techniques are tracer velocities, which are then assumed to be identical to the fluid velocities. This talk describes MnPIV results using tracers with diameters ranging from 40 nm to 100 nm for otherwise identical flows; changing tracer size should impact both electrostatic interactions and Brownian effects. Results are presented on how tracer size affects near-wall particle and velocity distributions in Poiseuille flow and electroosmotic (*i.e.*, electrokinetically driven) flow, with the additional complication of particle electrophoresis, in channels with a minimum cross-sectional dimension of about 40 μm.

[1]Supported by NSF
[2]Also Keio University

11:53AM PF.00002 Depth of correlation reduction due to out-of-plane shear in microscopic PIV , MICHAEL OLSEN, Iowa State University — The effects of out-of-plane shear on the depth of correlation in microscopic particle image velocimetry (microPIV) were analyzed by deriving an analytical model of microPIV interrogation for flowfields containing velocity gradients in the out-of-plane direction. The model is derived using a Taylor series approximation, and is therefore most accurate in the limit of small shear, but it does provide valuable insights. The model shows that out-of-plane velocity gradients reduce the depth of correlation compared to flowfields without gradients, but this decrease in depth of correlation is smaller than the increase in depth of correlation for a flowfield containing only in-plane velocity gradients. By combining the analysis for flows with in- plane gradients and with the analysis for flows with out-of-plane gradients, an equation for depth of correlation for a flowfield containing both in-plane and out-of-plane velocity gradients was derived. This equation suggests that unless the out-of-plane gradients are significantly larger than than the in-plane gradients, the effect on the depth of correlation due to the out-of-plane gradients is negligible, and the depth of correlation can be very closely approximated by calculating the depth of correlation using only the in-plane velocity gradients.

12:06PM PF.00003 Extending micro-PIV to unsteady flows and real time 3D particle tracking using spinning disk confocal microscopy[1] , STEVEN KLEIN, Mechanical Engineering, Arizona State University, JONATHAN POSNER, Mechanical Engineering, Chemical Engineering, Arizona State University — A high speed micro particle imaging velocimetry system based on a spinning disk confocal microscope is presented. The confocal system uses a spinning disk with a series of pinholes arranged in an Archimedes spiral to optically section the sample, rejecting light originating from out of focus particles. Unsteady flows require two frame cross correlations and can be better resolved using the increased correlation signal to noise ratio provided by the confocal system, resulting in less erroneous vectors for the same validation criteria and illumination intensity. By utilizing a high speed CMOS camera we can obtain vector fields at up to 2500 Hz, which is ultimately limited by the speed of rotating pinhole disk. Volumetric scans of fluorescently labeled microspheres are acquired by rapid translation of the high numerical aperture objective using 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 flows and cellular processes.

[1]This work was sponsored by an NSF CAREER Award (CBET-0747917) with William Wendell Schultz as grant monitor.

12:19PM PF.00004 Studies on Rheology of E-printing Inks by μ-PIV in Microchannels , YOUNG-SIK JANG, SIMON SONG, Hanyang Univ. — Using printing technologies for electronic circuits, such as antennas for radio frequency identification (RFID) chips, has been paid attention to recently in order to reduce production costs. In general, E-printing inks used for printed electronics have non-Newtonian properties because they contain metallic particles. Thus, it is important to investigate rheological behaviors of E-printing inks and suggest proper rheological models for developing printing devices for printed electronics. Also, the rheological models are necessary to accurately predict ink behaviors using CFD. However, classic methods to study rheological models are somewhat irrelevant since they require the mass consumption of expensive E-printing inks. Thus, to study rheological models suitable for commercial E-printing inks, we use microfluidic chips that only requires nascent E-printing inks. We measured flow velocities using μPIV and pressure drops along the microchannel to determine a relationship between stress and strain rate of ink flows. We found that the E-printing inks exhibit shear-thinning behaviors. In the presentation, we will propose rheology models suitable for the E-printing inks.

12:32PM PF.00005 Imaging the liquid film layer of slug flow within a microreactor[1] , SHAHRAM POUYA, MANOOCHEHR KOOCHESFAHANI, Michigan State University, ANDREW GREYTAK, DANIEL NOCERA, MOUNGI BAWENDI, VICKI DYDEK, KLAVS JENSEN, Massachusetts Institute of Technology — Segmented gas-liquid microreactors have gained attraction for high throughput material synthesis and sample processing in chemistry and biotechnology. The performance of the segmented gas-liquid microfluidic reactor derives from the uniformity of the gas-liquid segment lengths and the mixing that occurs within the liquid segment confined between gas slugs. The mixing process is a consequence of the recirculating flow that is set up within the liquid slugs. An important aspect of this flow geometry is that the liquid segments are not completely isolated but interconnected through a thin liquid film. Therefore, the behavior of the film layer and the flow field within the slugs are of great importance in hydrodynamics of the microreactor flow and improving the efficiency of such reactors. We present preliminary results of imaging, with quantum dots, the thin film layer surrounding the gas bubbles. The results are presented for stable slug flow of Ethanol/Nitrogen within a PDMS microreactor with channel size of 300$\times$250 micron.

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

12:45PM PF.00006 A simple microfluidc method for rapid generation of long-range material gradients in microchannels[1] , MATTHEW HANCOCK, YANAN DU, JINKANG HE, JOSE URIBE-VILLA, ALI KHADEMHOSSEINI, Harvard Medical School — The ability to recreate the heterogeneity of cellular environments is a major challenge for investigating cell- material interaction and for developing biomimetic materials for tissue engineering. Here we present a simple fluidic method for rapidly generating 2-3 cm gradients of biomolecules, polymers, microbeads, and cells. A polymer hydrogel gradient and a composite material with a cross-gradient of hyaluronic acid (HA) and gelatin in different ratios were generated with continuous variations in material properties. The approach relies on hydrodynamic stretching of the concentration profile and, for molecular species, enhanced lateral molecular diffusion. Faster gradient growth occurs for large Peclet numbers, outside the Taylor-Aris parameter range. Computational simulations and user-friendly power law formulas provide estimates of gradient growth for a wide range of Peclet numbers and channel cross-sections. Our microfluidic platform is limited to a rectangular PDMS microchannel and a syringe pump and should be accessible to a broad range of experimenters in the materials science and biomedical fields.

[1]Research supported by the US Army Engineer Research and Development Center, the Institute for Soldier Nanotechnology, NIH, the Coulter Foundation and the Draper Laboratory.

12:58PM PF.00007 Modeling the combined effect of surface roughness and shear rate on slip flow of simple fluids , ANOOSHEH NIAVARANI, NIKOLAI PRIEZJEV, Michigan State University — Molecular dynamics (MD) and continuum simulations are carried out to investigate the influence of shear rate and surface roughness on slip flow of a Newtonian fluid. The nonlinear shear-rate dependence of the intrinsic slip length in the flow over an atomically flat surface is computed by MD simulations. We describe laminar flow away from a curved boundary by means of the effective slip length defined with respect to the mean height of the surface roughness. Both the magnitude of the effective slip length and the slope of its rate-dependence are significantly reduced in the presence of periodic surface roughness. We then numerically solve the Navier-Stokes equation for the flow over the rough surface using the rate-dependent intrinsic slip length as a local boundary condition. Continuum simulations reproduce the behavior of the effective slip length obtained from MD simulations at low shear rates. The slight discrepancy between MD and continuum results at high shear rates is explained by examination of the local velocity profiles and pressure distribution along the wavy surface. We found that in the region where the curved boundary faces the mainstream flow, the local slip is suppressed due to the increase in pressure.

1:11PM PF.00008 Shear flow over arbitrary periodic surfaces , KEN KAMRIN, Harvard, HOWARD STONE, Princeton, MARTIN BAZANT, MIT — In a variety of applications, notably microfluidics, slip-based boundary conditions have been sought to characterize fluid flow over patterned surfaces. This work focuses on shear flows of Stokes fluid over surfaces with small height fluctuations and/or fluctuating Navier slip properties. Our goal is to derive a general formula to determine the "effective slip" in terms of surface properties and the applied shear stress. We show that the slip and the applied stress are always related linearly through a slip matrix, representing a tensorial mobility. The method of domain perturbation is then used to deduce an approximate formula for the slip matrix. We use the formula to determine optimal surface shapes and the effect of random fluctuations on fluid slip.

1:24PM PF.00009 Navier-slip and other viscous flow boundary conditions using penalty methods[1] , YVONNE STOKES, The University of Adelaide, GRAHAM CAREY, University of Texas at Austin — With the advent of microscale and nanoscale devices, the Navier-slip boundary condition as a macroscale model of fluid behaviour at a solid wall has seen renewed interest. The penalty concept and variational formulation are extended here to treat partial slip, free surface, contact and related boundary conditions for boundaries of general shape in viscous flow simulation. We analyse the penalty partial-slip formulation and relate it to the classical Navier-slip condition. A variant of the same penalty scheme also allows partial penetration through a boundary and, hence, the implementation of porous wall boundaries with leakage or "blowing and suction." Finally the scheme can be employed in a novel way to handle moving contact lines.

[1]This research was supported by Australian Research Council grant DP0450047, a University of Adelaide Special Studies Program grant, and the Curran Chair and ICES faculty visitor fellowship program at the University of Texas at Austin.

1:37PM PF.00010 A hybrid simulation of the atomistic-continuum methods for fluid flows on superhydrophobic surfaces , GUOWEI HE, QIANG LI, LNM, Institute of Mechanics, Chinese Academy of Sciences — It has been found from recent experiments that the fluid flows on superhydrophobic surfaces could have the slip length as large as the order of a micro-meter. The superhydrophobic surfaces can be achieved by patterning roughness on hydrophobic surfaces. In the present paper, an atomistic-continuum hybrid approach is developed to simulate the Couette flows over superhydrophobic surfaces, in which a molecular dynamics simulation is used to a small region near the superhydrophobic surface where the continuum assumption is not valid and the Navier-Stokes equations are used in a large region for bulk flows where the continuum assumption does hold. These two descriptions are coupled using the dynamic coupling model in the overlap region to ensure the momentum continuity. The hybrid simulation predicts a superhydrophobic state with the large slip lengths which cannot be obtained by molecular dynamics simulation alone.

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PG Free Surface Flows III 101G

11:40AM PG.00001 Numerical study of the convection induced by evaporation in cylindrical geometry , CLAUDINE DELCARTE, BENOÎT TROUETTE, Univ. Paris-Sud 11, LIMSI-CNRS, ERIC CHÉNIER, Univ. Paris-Est, MSME-CNRS, BÉATRICE GUERRIER, FAST-CNRS — During very short times, at the beginning of the drying of a polymer/solvent solution contained in a cylindrical crucible, experimental results have shown that the origin of convective cells is essentially linked to bouyancy- and/or surface tension-driven instabilities. In order to understand the relative importance of the two mechanisms, a 3D numerical study is performed. Convection is considered as significant when the Peclet number value (Pe) is greater than 1. The evolution of Pe as a function of the initial perturbation is intended to explore the transient character of the problem. The impact of the viscosity and thickness of the fluid on the convective regime and instability thresholds will also be presented and a comparison with experimental data performed. Convective patterns during the quasi-steady regime (slow evolution of the cells) will be shown for various aspect ratios.

11:53AM PG.00002 Experimental and Numerical Investigation of Convective Dominated Capillary Channel Flow in Microgravity[1] , JOERG KLATTE, MICHAEL DREYER, ZARM - Center of Applied Space Technology and Microgravity — In this work we investigated experimentally and numerically liquid flows through open capillary channels under microgravity conditions. The experimental investigations focus on the free surface contour and the maximum flow rate through the channel. Due to convective and viscous momentum transport the pressure along the flow path of the liquid decreases and causes the collapse of the free surface. This stability limit depends on the geometry of the channel and the properties of the liquid. We present an experimental setup which is used in the low gravity environment of the Bremen Drop Tower. High-Resolution Experiments with convective dominated systems have been performed where the flow rate was increased up to the maximum value. In comparison to this we present unique three-dimensional computations to determine important characteristics of the flow, such as the free surface shape and the limiting flow rate. The good agreement validates the capabilities of the numerical solver.

[1]The support for this research by the German Federal Ministry of Education and Research (BMBF) through the German Aerospace Center (DLR) under grant number 50WM0535/845 is gratefully acknowledged.

12:06PM PG.00003 Investigation of capillary flow in channels with polygonal cross sections: Simulations and experiments[1]

, JIANSHENG FENG[2], JONATHAN ROTHSTEIN[3], Univ. Massachusetts Amherst — A series of simulations and experiments will be presented which systematically investigate the flows in channels having polygonal cross sections through capillary action. Specific attention is focused on the evolution and shape of the resulting menisci. Simulations are preformed using Surface Evolver in which several important parameters were varied including the number of sides, the characteristic length scale, the contact angle, and the curvature of the channel corners. The results are used to guide and compare with corresponding experiments of capillary force lithography (CFL). The elastomeric molds used in CFL are made by casting PDMS on a rigid patterned master previously fabricated by electron beam lithography combining with reactive ion etching to incorporate precise patterns of polygonal capillaries with the size of around 100nm. Products of CFL are characterized by atomic force microscopy as well as scanning electron microscopy. The results demonstrate that it is possible and practical to fabricate hierarchic structures of sub-100nm features on top of sub-micron patterns.

[1]Supported by UMass MRSEC and CHM.
[2]Graduate Research Assistant
[3]Principal Instructor

12:19PM PG.00004 Multifunctional superhydrophobic coatings for large area applications

, CONSTANTINE MEGARIDIS, THOMAS SCHUTZIUS, ARINDAM DAS, MANISH TIWARI, University of Illinois at Chicago, ILKER BAYER, University of Illinois at Urbana-Champaign — Formulation of flexible superhydrophobic coatings (water droplet contact angles above 150 deg and roll-off angles below 10 deg) with high durability and electrical conductivity, and their fabrication using scalable techniques is a major challenge. The current work lays their foundation using solution processed polymer nanocomposites. Carefully selected polymer(s) are used to disperse filler particles and the dispersions are applied by spraying process. The filler particle size, surface energy and other functionalities are varied to produce the coatings. Sub-micron poly(tetrafluoroethylene) (PTFE) particles and carbon black or other nanoparticles are jointly used to obtain hierarchical morphology (micro-to-nanoscale roughness) and superhydrophobicity. As examples, firstly, acrylonitrile-co-butadiene rubber based nanocomposites are shown to maintain superhydrophobicity up to 200% linear and for 100 cycles of reversible 0 to 100% uniaxial stretching. Secondly, poly(vinylidene fluoride) and poly(methyl methacrylate) blend based nanocomposites containing carbon nanofibers are demonstrated as superhydrophobic coatings with electrical conductivity up to 300 S/m.

12:32PM PG.00005 Dynamics of complete wetting liquid under evaporation[1]

, CHI-TUONG PHAM, Laboratoire d'Informatique pour la Mécanique et les Sciences de l 'Ingénieur, UPR 3251 CNRS & Université Paris-Sud 11, Orsay, France, GUILLAUME BERTELOOT[2], Laboratoire Matière et Systèmes Complexes, UMR 7057 CNRS & Un iversité Paris Diderot, Paris, France, FRANÇOIS LEQUEUX, Laboratoire Physico-chimie des Polymères et Milieux Dispersés, UMR 7615 CNRS, ESPCI, Paris, France, LAURENT LIMAT, Laboratoire Matière et Systèmes Complexes, UMR 7057 CNRS & Un iversité Paris Diderot, Paris, France — We study the dynamics of a contact line under evaporation and complete wetting conditions taking into account the divergent nature of evaporation near the border of the liquid, as evidenced by Deegan et al. [Nature **389**, 827]. The model we propose shows the existence of a precursor film at the edge of the liquid. The length of the precursor film is controlled by Hamacker constant and evaporative flux. Past the precursor film, Tanner's law is generalized accounting for evaporative effects.

[1]The authors thank ANR DEPSEC 05-BLAN-0056-01 for fundings. G. B. acknowledges funding from Direction Générale de l'Armement (DGA)
[2]Laboratoire Physico-chimie des Polymères et Milieux Dispersés, UMR 7615 CNRS, ESPCI, Paris, France

12:45PM PG.00006 Micron-scale measurements of the flow field near a moving contact line

, BIAN QIAN, KENNETH BREUER, Brown University — It has long been known that a continuum hydrodynamic description using a no-slip boundary condition breaks down near a moving contact line. Theoretical models including microscopic effects, such as velocity slip and a diffuse interface, have been proposed to relieve the contact line singularity. Although experimental testing of the theoretical models has been attempted by measuring the apparent dynamic contact angle, few efforts have been made to map the flow field close to a moving contact line. We experimentally investigated the flow motion near the moving contact line of a liquid bridge, which is trapped between a stationary hanging rod and a glass substrate which can be moved at a controlled speed. The flow field was seeded with nano-scale fluorescent particles visualized using both flood illumination and evanescent wave (TIRF). The motion was captured using a high speed camera equipped with a high-magnification microscopic objective. These experimental arrangements enable to resolve the flow field within 10 microns of the contact line and as close as 100nm above the moving substrate. The characteristics of the flow, including slip lengths as a function of distance from the contact line can be calculated from the flow field.

12:58PM PG.00007 Experimental Investigation of the Effects of Surface Conditions on Natural Convection-Driven Evaporation[1]

, S.M. BOWER, J.R. SAYLOR, Clemson University, Dept. Mechanical Engineering — Presented are the results from an experimental investigation of the effects of surface conditions at an air/water interface on transport phenomena within the context of natural convection-driven evaporation. Experiments were conducted using tanks of heated water under several different surface conditions: 1) contamination with an oleyl alcohol monolayer, 2) contamination with a stearic acid monolayer, and 3) "clean" or surfactant-free. These surface conditions create the following hydrodynamic boundary conditions: 1) constant elasticity, 2) no-slip, and 3) shear-free. The effect of these boundary conditions on evaporation and air-side natural convection heat transfer is presented via the power law relationships between the Sherwood and Rayleigh numbers (for evaporation) and the Nusselt and Rayleigh numbers (for natural convection heat transfer). Additionally, infrared imagery of the water surface was collected during these experiments, yielding qualitative information on the effect of these boundary conditions on the flow near the interface. Few studies exist in which the effects of surface conditions on interfacial heat and mass transfer are investigated, making this work particularly relevant.

[1]Support from the National Science Foundation is gratefully acknowledged.

1:11PM PG.00008 The existence of longitudinal vortices in the flow of air above an air/water interface[1]

, J. KOU, J.R. SAYLOR, Clemson University — Many researchers have observed the formation of longitudinal vortices in boundary layers developing over heated solid surfaces. In the present work, such vortices were observed in an air boundary layer developing over a heated water surface. The existence of these vortices was documented via infrared imaging of the water surface, which showed a consistent pattern of hot and cold streaks, coinciding with the vortex position. These vortices were also visualized through smoke injected into the air-side flow. The onset position X_c and lateral vortex spacing λ were investigated for a range of wind speeds (0.1 - 1 m/s) and air/water temperature differences (26 - 42 °C). Plots of X_c/λ versus the Reynolds number exhibit power-law behavior similar to that of prior work on boundary layers over heated solid surfaces. However, plots of X_c/λ versus the Grashof number show significant differences from the power-law behavior observed for heated solid plates. A theory explaining the similarity and difference between the present results and those for heated solid plates is discussed which is based on differences in the thermal boundary conditions.

[1]Support from the Department of Energy and the National Science Foundation is gratefully acknowledged.

1:24PM PG.00009 On hydrostatic free surface flows[1] , MICHAEL RENARDY, Virginia Tech — We consider the hydrostatic Euler equations in a strip bounded by two free surfaces. We show that the initial value problem is well-posed only if the horizontal velocity is uniform in every cross section of the strip.

[1]Supported by the National Science Foundation.

1:37PM PG.00010 Stretching bridges and bubbles: The effect of air bubbles on liquid transfer , SHAWN DODDS, University of Minnesota, MARCIO S. CARVALHO, PUC-Rio, SATISH KUMAR, University of Minnesota — The transfer of liquid from one surface to another is important in a wide variety of both natural and industrial settings. A potential complication in an application such as printing is the entrapment of air bubbles, which can alter the amount of liquid transferred to the substrate, and can produce defects if the bubbles do not burst before drying. To better understand this effect, we use flow visualization to study the stretching of liquid bridges at low capillary numbers. The bridges are visualized from beneath the drop to track the motion of the contact lines, and from beside the drop to enable image analysis to quantify liquid transfer. The dynamics of the outer gas-liquid interface, between the drop and the surrounding air, are found to be the same during stretching both with and without a bubble; thus, the volume of fluid (liquid or liquid plus bubble) transferred to the moving surface is the same in both cases, regardless of the dynamics of the bubble. Therefore, the liquid transfer will increase if the bubble remains on the stationary surface after breakup, or decrease if it transfers to the moving surface. The effect of the wettability of the surfaces on the breakup behavior of the liquid bridge is also discussed.

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PH Drops VIII: Contact Lines 101H

11:40AM PH.00001 Capillarity driven contact line motion in cyclic bridge-drop grab-release events , HENRIK VAN LENGERICH, PAUL STEEN, Cornell University — Motivated by a reversible adhesion device which uses capillary forces to adhere to a substrate, we study the mechanical work done in a grab-release cycle. That is, the volume of a drop is increased until it grabs the substrate and forms a bridge and then shrunk until it goes unstable and releases from the substrate and forms a drop again. In the instant that a drop becomes a bridge (or vice versa) no work is done on the system, however, energy is dissipated due to the decrease in interfacial energy. This dissipation can be compared with the mechanical dissipation based on the fluid flow. For viscous fluids, a wedge model shows that most of the dissipation occurs in the vicinity of the contact line. The thermodynamic dissipation is compared with that expected in the fluid without the need of static contact angle or slip length.

11:53AM PH.00002 Two dimensional droplets under partially wetting conditions: New analytical solutions , J.M. GOMBA, Facultad de Ciencias Exactas - Universidad Nacional del Centro - Argentina, G.M. HOMSY, Dept. Mech. Engr, UCSB, USA — We present new analytical solutions that describe the static shape of a two-dimensional droplet in equilibrium with a surrounding thin film on a solid substrate. We model a partially-wetting liquid by a disjoining-conjoining pressure description of intermolecular forces between the liquid and solid substrate. The profile involves three regions: (i) a droplet core, (ii) a thin film region and (iii) an intermediate contact line region. The description of the droplet shape in these regions is usually carried out by series expansions or numerical solutions. In contrast, here we derive new fully- analytical solutions for the shape of the droplet, the cross sectional area, the half width and the maximum curvature. We also study the effects of the size of the droplet on the apparent contact angle. We find that for nanodroplets the contact angle follows a power law dependence with the cross sectional area, reaching a well defined value for larger droplets.

12:06PM PH.00003 Modeling of dynamic wetting far from equilibrium , ANDREAS CARLSON, MINH DO-QUANG, GUSTAV AMBERG, Linné Flow Center, Department of Mechanics, The Royal Institute of Technology — Moving contact lines in dynamic wetting phenomena have been studied extensively for several decades, nonetheless, the physics that drive such processes are not fully understood. Continuum mathematical models for such phenomena often rely on ad-hoc physical assumptions or simplifications. We present here numerical simulations of dynamic wetting far from equilibrium based on a free energy formulation. A direct qualitative and quantitative match with the experiments by [Bird, J. C., S. Mandre, and H. A. Stone, 2008, PHYSICAL REVIEW LETTERS 100(23)] is shown. To correctly capture the dynamics of rapid wetting, we demonstrate that it is crucial to account for non-equilibrium at the contact line. A term in the boundary condition at the solid surface, that naturally arises in the phase field theory, is interpreted as allowing for the establishment of a local structure in the immediate vicinity of the contact line. Besides one universal non-dimensional number, that is determined here, the model as presented has no free parameters.

12:19PM PH.00004 Wall energy relaxation in Cahn-Hilliard model for moving contact lines , PENGTAO YUE, Virginia Polytechnic Institute and State University, JAMES FENG, University of British Columbia — Contact angle in the Cahn-Hilliard model is determined by wall energy. The finite-rate relaxation of this wall energy results in a dynamic contact angle which differs from the static one. According to our numerical simulation, the wall energy relaxation is crucial to the successful fitting of experimental data with a numerically manageable slip length, which could be two orders of magnitude larger than the physical one. Through a simple analysis, we establish a relationship between the dynamic contact angle and the capillary number, which is verified by our numerical simulation. We further show that this relationship is consistent with Cox's hydrodynamic model. In a sense, the wall energy relaxation coarse-grains an area surrounding the contact line into a "slip region" while keeping the apparent contact angle outside the region unchanged. In the end, we show some new results on drop spreading.

12:32PM PH.00005 Evolution of the Diffusive Precursor Film of a Wetting Fluid at the Vicinity of the Moving Contact Line , ANNA HOANG, PIROUZ KAVEHPOUR, UCLA — For wetting fluids, a microscopic film, which is known as the precursor film, exists at the front of the moving contact line. The structure of this thin film has been studied theoretically, but previous experimental investigations were limited by the resolution of the measurement system (lateral or vertical). We studied the evolution of the profile of a spreading droplet near the moving contact line using total internal reflection fluorescence microscopy (TIR-FM). Our technique provides the lateral resolution and dynamic range required to capture the features of the macroscopic drop (spherical cap), wedge region, and precursor film within a single experiment. The dynamic characteristics of the precursor film are in good agreement with the theoretical results.

12:45PM PH.00006 Laser Speckle Drop Profilometry Validation and Measurement of Contact Angle Variation with Surface Roughness , JASON SCHMUCKER, EDWARD WHITE, JOSHUA OSTERHOUT, Texas A&M University — A non-intrusive technique has been developed that measures full-field instantaneous interface shapes of unsteady droplets on rough surfaces. Illumination of a rough surface by a collimated laser forms a speckle pattern at the solid surface that is subsequently deformed by refraction at the drop interface, encoding information about the surface height and gradient. Computer algorithms analyze the resulting images to identify the interface shape, contact line location and contact angles about the contact line. This is achieved through a minimization of the mean-squared error between the measured speckle deformation and that of the reconstructed drop using simulated annealing. Extensive validation efforts demonstrate the technique's effectiveness on aluminum, copper, and stainless steel surfaces when the surface roughness is micron scale. Preliminary experiments provide data on how contact angle variations about a single drop's contact line and between different droplets depends on surface roughness on the various surfaces.

12:58PM PH.00007 Investigating Contact Angle Forces with Traction Force Microscopy , ELIZABETH JERISON, ERIC DUFRESNE, Yale University — Although the classic Young's relation indicates that the contact angle between a drop and substrate is a constant material property, observation of an evaporating water drop on solid PDMS reveals four phases of contact angle dynamics: spreading, pinning, isotropic contraction, and contraction with a decreasing contact angle. Only in the third phase does the contact angle remain constant. We use traction force microscopy to visualize the forces exerted by the drop contact line on the substrate, with the goal of explaining this contact-angle anomaly.

1:11PM PH.00008 Oscillations of a viscous drop under spherical-belt constraint , JOSHUA BOSTWICK, PAUL STEEN, Cornell University — The motion of constrained liquid-gas interfaces is important in a variety of applications. We study the linear oscillations of a viscous liquid drop immersed in an immiscible fluid and constrained by an axisymmetric spherical "belt." The belt is a rigid spherical band delimited by two latitudinal circles. Unlike the unconstrained(Rayleigh) problem, the liquid boundary is the union of a surface of support and two disconnected free surfaces, the latter coupled by the incompressibility condition and allowed to "communicate" across the constraint. A modified set of shear boundary conditions is introduced to address the transition from free to supported surfaces along the drop interface. This formalism allows mode shapes with discontinuous contact angle across a pinned circle-of-contact constraint, a limiting case which is consistent with observation. As the size of the constraint increases from a pinned circle of contact, the mode shapes are shown to qualitatively change their character by increasing the number of nodes of the corresponding eigenfunction, while preserving the numerical ordering of the eigenfrequencies at a critical belt size.

1:24PM PH.00009 The Vibration of an Inviscid Incompressible Sessile Drop , MARC SMITH, Georgia Institute of Technology — The fundamental frequencies and normal modes of vibration of a sessile drop supported on a horizontal planar surface are found using an integrated analytical and numerical technique. Spherical coordinates are used to describe the interface shape, but the potential flow field inside the drop is computed numerically using the finite element method. The numerical velocity potentials at the interface for both the fluid inside the drop and outside are fitted using a Legendre series. When these series are combined in the interfacial normal-stress balance the result is a linear eigenvalue problem that is solved numerically. Results will be presented for sessile drops with different contact angles without gravity and compared to experimental data. This technique can also be extended to sessile drops with gravity, in which the drop shape is flattened, and to substrate geometries that are not planar, such as a drop in a shallow cavity or hole.

1:37PM PH.00010 The Shape of a Sessile Drop , BHARADWAJ PRABHALA, MAHESH PANCHAGNULA, SRIKANTH VEDANTAM, VENKAT SUBRAMANIAN — Contact angle is an important parameter which characterizes the interaction between the liquid and solid surface. Based upon the contact angle solid substrates are primarily classified as hydrophilic, hydrophobic and super hydrophobic substrates. In this study, we investigate the dependence of the local contact angles on the configuration of the contact line shape of a sessile liquid drop on a solid substrate. We use the numerical algorithm 'Surface Evolver' which is an interactive program for the study of surfaces shaped by surface tension, gravitational and other energies. The algorithm calculates the velocity at each vertex and the vertex is moved in the direction such that the surface evolves towards a minimum energy. The shapes and the energies of the drop are computed using the Surface Evolver. An analytical solution based on perturbation expansion is developed to predict the shape of the sessile drop for a given contact line description. The shape of the contact line is also varied in the same manner and the exact drop shape is computed in Surface Evolver. The Root Mean Square error is calculated by comparing the radius at all the vertices between the analytical and numerical approaches for varying contact angles and amplitudes of undulations. We show that the applicability of the analytical solution is quite widespread.

Tuesday, November 24, 2009 11:40AM - 1:24PM –
Session PJ Waves II 101I

11:40AM PJ.00001 Evolution of depressed and elevated tsunami waves , H.J.S. FERNANDO, Arizona State University (ASU), C.A. KLETTNER, University College London (UCL), S. BALASUBRAMANIAN, Los Alamos National Laboratory, J.C.R. HUNT, UCL, S.I. VOROPAYEV, ASU, I. EAMES, UCL — Tsunamis differ greatly between positive (elevated) and negative (depressed) waves. These differences are explained using a hydraulic model based on the conservation of impulse. Laboratory experiments of depression waves, conducted using a novel wave-maker, are compared with model predictions. Over a sloping beach, they have a nearly constant V-shaped depression trailed by a growing lambda- shaped positive wave. The shoreline recedes (draw-down) over a significant distance, caused by shoreward water being drawn toward the V-shaped depression. When the lambda- shaped positive wave breaks (spills), an energetic hydraulic bore develops and moves up the beach. The model leads to general formulae for wave slopes, draw-down and run-up. The run up of negative waves can be larger or smaller than positive waves, depending on the wave amplitude and beach parameters. The predictions are consistent with videos and photographs taken during the 2004 Sumatra tsunami. Using the amplitude data in the new tsunami warning systems, the properties of tsunamis on beaches could be estimated in real time using the present work, thus improving emergency response strategies.

11:53AM PJ.00002 Harmonic Generation of Internal Waves Reflected from a Slope , BRUCE RODENBORN, D. KIEFER, H.P. ZHANG, HARRY L. SWINNEY, Center for Nonlinear Dynamics, University of Texas at Austin, USA — The reflection of oceanic internal waves from bottom topography can generate harmonics and cause mixing that could be important in sustaining the meridional overturning ciruclation. Internal wave reflection is often treated as a linear or a weakly nonlinear inviscid problem[1]. Under these assumptions, and for a linearly stratified fluid, Thorpe[2] and Tabaei et al.[3] derived predictions for the boundary angle where second harmonic generation should be most intense. We have conducted laboratory experiments and two-dimensional numerical simulations of the Navier-Stokes equation in the Boussinesq approximation to test these predictions. The results from experiments and simulations are in agreement but differ from both theories, except for very low intensity incoming waves. However, we obtain an empirical geometric relationship between the wave beam and boundary angles that predicts a condition for maximal second harmonic generation, and that agrees with the results from our experiment and simulation.

[1]T. Dauxois and W.R. Young, J. Fluid Mech. **390**, 271-295 (1999)
[2]S. A. Thorpe, J. Fluid Mech., **178**, 279-302 (1987)
[3]A. Tabaei, T. R. Akylas and K. G. Lamb, J. Fluid Mech. **526**, 217-243 (2005)

12:06PM PJ.00003 Deep Ocean Wave Cancellation Using a Cycloidal Turbine , STEFAN SIEGEL, TIGER JEANS, THOMAS MCLAUGHLIN, US Air Force Academy — We investigate the use of a cycloidal turbine for deep ocean wave termination 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, the shaft is parallel to the wave crests, and the turbine operates in sync with the wave frequency by means of feedback control. The approach differs from traditional approaches in that it is a lift based system and therefore has the potential to be more efficient than existing drag based converters. It also allows for feathering of the blades in order to survive storms. We present two-dimensional inviscid results of potential flow simulations modeling the turbine blades as single point vortices of constant circulation rotating under a linearized free water surface. With suitable parameter choices for the turbine radius, blade number, submersion depth and airfoil circulation up to 97% of the incoming deep ocean Airy wave energy can be converted to shaft power. For a typical North Atlantic deep ocean wave this corresponds to 100 kW of power per meter of wave crest. The remaining energy is lost to harmonic waves travelling both in the up- and down wave directions.

12:19PM PJ.00004 ABSTRACT WITHDRAWN –

12:32PM PJ.00005 Synchronous Sloshing in a Free Container , ANDRZEJ HERCZYNSKI, Boston College, PATRICK WEIDMAN, University of Colorado, Boulder — A standing wave in a container partially filled with liquid and free to move along one horizontal axis is analyzed. Interaction of the sloshing liquid with the container drives the acceleration of the vessel, which oscillates back-and-forth, out-of-phase with the liquid oscillations. Linearized shallow water theory is employed to obtain lowest-mode frequencies for rectangular and cylindrical containers. Validity of the results require $h/\lambda << 1$ where h is the fluid depth and λ is the sloshing wavelength, and $\eta_o/h << 1$ where η_o is the wave amplitude. Experiments using containers with water, supported on a low-friction cart constrained to move in one dimension, reveal excellent agreement with theory up to a certain liquid depth corresponding to the shallow-water limit. Beyond that critical depth, the observed frequencies of oscillation are lower than linear predictions and thus full potential theory is required.

12:45PM PJ.00006 Numerical and Experimental Study of the Dynamics of Imploding Hydraulic Jumps , AMY-LEE GUNTER, OTHMAN SMADI, Concordia University, CHARLES KIYANDA, University of Illinois at Urbana-Champaign, LYES KADEM, HOI DICK NG, Concordia University — The dynamics of imploding hydraulic jumps is investigated in this study. Experimental and numerical studies are performed to identify the critical conditions at which circular shallow water waves can be produced and amplify as they propagate toward the center without wave breaking. These conditions enable water waves to behave analogously to gaseous shock waves through the hydraulic analogy. The stability of the imploding jumps is also analyzed by introducing obstacles in the path of the implosion. Experimentally, a gate-type water table is constructed and the creation of a circular converging hydraulic jump is achieved by retracting the gate which separate two volumes of water by mean of three pneumatic pistons. A CCD camera is used to visualize the dynamics of the implosion. The acquired images are processed on Matlab using an image processing toolbox based algorithm which detects the shape and trajectory of the imploding wave. To compare the characteristics of the imploding jump and the mechanism of wave breaking, numerical simulations using Volume of Fluid (VOF) and Smoothed Particles Hydrodynamics (SPH) methods are performed. The experimental and numerical results are compared with the Chester-Chisnell-Whitham (CCW) approximate solution of the shallow water wave equations.

12:58PM PJ.00007 Similarity solution for strong exploding shock waves in water , JIA ZHANG, HAO LIN, Rutgers University — In single bubble sonoluminescence, strong bubble oscillations often lead to the generation of spherical shock waves which expand outward in the surrounding media. These shock waves offer valuable information to help diagnose the cavitation events at the center. In this work, the similarity behavior of these waves is investigated using analyses and simulations. A fitted analytical equation of state (EOS) is first extracted from a tabular EOS for water (LANL), and then incorporated into the Euler equations. A similarity solution for the flow variables behind a shock wave is derived following the theory by Taylor (1950). Numerical simulations are performed using the original tabular EOS, to compare with and validate the analytical solution. The result indicates that the shock wave propagation can be divided into three regimes. 1. A strong-shock regime where the wave-front location scales to the 2/5 power of time, and the similarity solution well-describes the flow dynamics. 2. A sonic regime where the shock wave weakens and propagates with the local speed of sound. 3. A transition regime connecting the two above. Based on these results, a time-of-flight theory is developed, from which the energy of the initial shock wave can be estimated given the arrival time of the shock wave-front at a defined detector location.

1:11PM PJ.00008 Do Waveless Ships Exist? , PHILIPPE TRINH, JON CHAPMAN, Oxford University, JEAN-MARC VANDEN-BROECK, University College London — Consider two-dimensional ideal and low-speed flow past a ship modeled as a semi-infinite body with constant draft. In the 1970s, on the basis of numerical evidence, it was conjectured that ships with a single front face will always generate a wake. Later in the 1980s, seemingly waveless ships with bulbous profiles were discovered. And finally, conflicting evidence in the 1990s suggested that the waves were in fact present, but simply too small to be recognized. In this talk, we will show how recent techniques in exponential asymptotics can be used in order to study the ship-wave problem. In particular, we will show how the formation of waves near a ship are a necessary consequence of singularities in ship's geometry, such as those corresponding to sharp corners or stagnation points. Finally, we will show how the theory can be applied in order to prove that certain ship profiles will or will not produce a wake in the low-speed limit.

Tuesday, November 24, 2009 11:40AM - 1:50PM –

Session PK Fluid-Driven Particle Flows 101J

11:40AM PK.00001 Effect of dynamic bed topography on turbulent flow structure in a gravel-bed river , ARVIND SINGH, FERNANDO PORTE-AGEL, EFI FOUFOULA-GEORGIOU, University of Minnesota — A series of flume experiments were conducted at St. Anthony Falls Laboratory, University of Minnesota to study the effect of bedform dynamics on the flow over a gravel bed comprised of a wide distribution of grain sizes. Instantaneous high-frequency velocities were sampled using ADV at a frequency of 200 Hz, while the simultaneous bed elevations were sampled using SONAR transducers at a frequency of 0.1 Hz. Spectral analysis of the measured velocity fluctuations reveals the existence of two distinct power-law scaling regimes. At high frequencies, an inertial subrange of turbulence with Kolmogorov scaling is observed. At low frequencies, another scaling regime with spectral slope of about -1.1 is found. This range is the signature of the evolving multi-scale bed topography on the near-bed velocity fluctuations. The two scaling ranges are separated by a spectral gap, i.e., a range of intermediate scales with no energy contribution. The high-frequency limit of the spectral gap corresponds to the integral scale of turbulence. The low-frequency end of the gap corresponds to the scale of the smallest bedforms identified by the velocity sensor, which depends on the position of the sensor. Comparison with spectral densities of bed elevations also shows that relatively low-resolution velocity measurements collected near the channel bed can be used to estimate the travel time of the largest bedforms.

11:53AM PK.00002 The effects of woody debris on streambed morphology: flume experiments on the spatial patterns of fluid flow and sediment transport around woody debris , VIVIAN LEUNG, DAVE MONTGOMERY, University of Washington — The interaction of woody debris, fluid flow and sediment transport in rivers creates local streambed morphology, such as large pools that are important fish habitat and sediment deposits that bury and stabilize wood. We present the results of 300 experimental runs characterizing the fluid flow field around individual stationary model wood on an immobile bed. Fluid flow patterns on the bottom boundary layer, where bedload sediment transport occurs, were visualized using solid dye crystals. We find that: 1) the presence of roots leads to greater areas of predicted sediment scour and deposition; 2) the amount of predicted sediment scour and deposition are exponentially related to the root cross-sectional area, oriented orthogonal to flow; 3) as root porosity decreases the amount of predicted sediment scour and deposition decrease and the predicted sediment deposit moves away from the roots. Ongoing sediment transport experiments, building on the fluid flow experiments, are investigating the volumes and spatial patterns of sediment scour and deposition around woody debris.

12:06PM PK.00003 ABSTRACT WITHDRAWN –

12:19PM PK.00004 Nonlinear convection in a mushy layer: Chimney spacing and optimal brine fluxes , ANDREW WELLS, STEVEN ORSZAG, JOHN WETTLAUFER, Yale University — The rapid solidification of any binary alloy leads to the formation of a chemically reactive porous medium, or mushy layer, comprised of a dendritic solid phase threaded by a concentrated fluid. An important geophysical example is sea ice, where solid ice crystals are separated by dense salty brine, the buoyancy-driven drainage of which has important implications for the ocean thermohaline circulation, and the long time bulk mechanical and electromagnetic properties of the sea ice matrix itself. It is known that convection and local dissolution lead to flow focusing in drainage channels devoid of solid, or chimneys. The spacing of chimneys and resulting brine fluxes evolve over time. We consider nonlinear convection within a mushy layer, applying a numerical model of directional solidification to investigate the spacing mechanism for fully developed chimneys. The resulting dynamics yields insight into the evolution of solute fluxes from growing mushy layers.

12:32PM PK.00005 Direct Numerical Simulation of Turbulent Flow over Sandy Rippled Beds , ALLISON PENKO, JOSEPH CALANTONI, Marine Geosciences Division, Naval Research Laboratory, DONALD SLINN, Dept. of Civil and Coastal Engineering, University of Florida — The presence of ripples on the seafloor affects the turbulent dynamics of bottom boundary layer (BBL) flow. The difference in the roughness length scales of a planar and rippled sand bed produces quantifiable differences in the turbulent BBL. A complete understanding of the effect of suspended sediment concentration on turbulence modulation is currently unknown. We use mixture theory to implement a three-dimensional BBL model that simulates the coupled interaction between the fluid and sediment. The mixture theory approach treats the fluid-sediment mixture as a single continuum with effective properties that parametrize the fluid-sediment and sediment-sediment interactions. We compare two-dimensional and three-dimensional simulations with existing laboratory measurements of fluid velocity and sediment concentration over rippled sand beds. We find that the vortex dynamics over sand ripples are highly three-dimensional. Two-dimensional flow simulations are inadequate for the numerical modeling of turbulent flow over sand ripples. We also find that suspended sediment concentration influences the production of turbulence; therefore, accurate simulation of turbulent flow over sandy beds must include an adequate description of fluid-sediment interactions.

12:45PM PK.00006 Experimental Studies on the Saltating Sand Particle Transport and Wind-Sand Interaction , WEI ZHANG, Iowa State University, SANG JOON LEE, Pohang University of Science & Technology, South Korea — Saltation is the major transport mode of wind-blown sand particles, accounting for about 75% of total sand transport through saltation, suspension and surface creep. The complex interactions among the saltating sand particles, the particles on the surface and the turbulent flow have not been fully understood owing to lack of experimental data. Various state-of-the-art flow measurement techniques were applied to comprehensively examine three different types of natural sand in a simulated atmospheric boundary layer. Firstly, digital high-speed photography was used to capture images of the saltating sand particles at 2000 frames per second, which resolved the particle motion adjacent to the sand bed surface. Secondly, instantaneous velocities of the saltating sand particles were extracted from the high-speed particle images using the particle tracking velocimetry (PTV). The particle resultant velocity, concentration and the stream-wise mass flux were evaluated as a function of height. Finally, the velocity fields of wind and wind-blown sand particles were simultaneously measured by using the PTV and the particle imaging velocimetry (PIV), respectively. This experimental study shed new lights on the complicated saltation motion, and will be helpful in enhancing formulation of theoretical models and development of effective control measures of wind erosion.

12:58PM PK.00007 Blowing in the wind: aeolian dunes in modulated gravity , WILLEM VAN DE WATER, DANIEL SNOUCK, Physics Department, Eindhoven University of Technology, PO Box 513, 5600 MB, Eindhoven, the Netherlands — Barchan dunes can be found in the desert under steady wind conditions. They translate in the direction of the wind while their shape remains unchanged. These remarkable natural patterns are the result of the interaction between sand and wind where the wind deposits the sand in heaps, which, in turn, change the properties of the turbulent wind. These crescent-shaped dunes have a minimal length in the order of ten meters, which renders laboratory experiments almost impossible. Their length scale is set by the details of the sand-wind interaction. In nature, smaller dunes do not evolve into the typical barchan shape. Our experimental approach produces dramatically scaled down dunes. The idea is to modulate gravity by vertical oscillation of the sand bed. Our tiny dunes travel in the turbulent boundary layer of an open windtunnel. Particle image velocimetry on their surface reveals the flux of creeping sand, while measurement of sand grains flying through the air using a high speed camera quantifies the key mechanism that moves sand by wind: saltation. We will contrast our findings with several theories that predict the shape of dunes on earth and other planets of our solar system.

1:11PM PK.00008 Theoretical and Experimental Investigation of Lunar and Martian Regolith Simulant Dynamic Response to Rocket Plume Impingement , JOHN BRANDENBURG, Orbital Technologies Corporation, ROBERT BEHRINGER[1], ABRAHAM CLARKE, Duke University, ORBITEC COLLABORATION, DUKE UNIVERSITY COLLABORATION — An investigation of rocket plume impingement on the regolith of the Moon and Mars is being conducted both theoretically and experimentally. Experimental results (1)and data from the Apollo landings inspired a theoretical model at ORBITEC : the ABL (Ablating Boundary Layer) model that assumes that regolith erosion and entrainment occurs in the thin boundary layer. The resulting crater streamlines itself with curve formed by extremization of the Lagrangian : L = (Z')^2+ Z^2 where Z(r) and Z(r)' are a depth variable and its radial derivative respectively. The actual depth profile z (r) in this model is derived from the formula z=Log (1+ Z/Z$_o$) where Z$_o$ is a constant. For light soils the model reduces to z$\sim$ Z/Z$_o$ and cantenary profiles result, exponential density profiles (2) give conoidal craters. (1) Experimental tests of the ABL model performed at Duke have shown good agreement. Further theoretical modeling and experimental data will be presented. (1) Metzger P., Lane, J., Immer C. and Clements, S. '6th International Conference on Case Histories in Geotechnicla Engineeering , Arlinton VA August 11-16, 2008. (2) Bresson L. M., Moran C. J., and Assoline, S. Soil Sci. Soc. of Am. Jou, 2004, vol. 68, 4, pp. 1169-1176.

[1]bob@phy.duke.edu

1:24PM PK.00009 Granular Crater Formation[1] , ABE CLARK, ROBERT BEHRINGER, Duke University, JOHN BRANDENBURG, ORBITEC — This project characterizes crater formation in a granular material by a jet of gas impinging on a granular material, such as a retro-rocket landing on the moon. We have constructed a 2D model of a planetary surface, which consists of a thin, clear box partially filled with granular materials (sand, lunar and Mars simulants...). A metal pipe connected to a tank of nitrogen gas via a solenoid valve is inserted into the top of the box to model the rocket. The results are recorded using high-speed video. We process these images and videos in order to test existing models and develop new ones for describing crater formation. A similar set-up has been used by Metzger et al.[2] We find that the long-time shape of the crater is consistent with a predicted catenary shape (Brandenburg). The depth and width of the crater both evolve logarithmically in time, suggesting an analogy to a description in terms of an activated process: $dD/dt = A\exp(-aD)$ (D is the crater depth, a and A constants). This model provides a useful context to understand the role of the jet speed, as characterized by the pressure used to drive the flow. The box width also plays an important role in setting the width of the crater.

[1]Work supported by USAF Contract #NNX09CF72P, and ORBITEX subcontract OTC-GS-0238-1.

[2]P. T. Metzger et al. Journal of Aerospace Engineering (2009)

1:37PM PK.00010 Euler-Lagrange Simulations of Particle Interactions with Coherent Vortices in Turbulent Boundary Layers , FERNANDO MORALES, Arizona State University, Tempe, AZ, IFTEKHAR NAQVI, Queens University, Kingston, Canada, KYLE SQUIRES, Arizona State University, Tempe, AZ, UGO PIOMELLI, Queens University, Kingston, Canada — The overarching interest of the current investigations is numerical modeling of particle entrainment and deposition near sandy beds as relevant to the problem of rotorcraft brownout. Numerical simulations are being performed using an Euler-Lagrange method. Solution of the incompressible gas-phase flow field is accomplished using a fractional-step numerical method; the particulate phase is advanced using Discrete Particle Simulation. The particular flow field of interest models a rotor wake and is comprised of coherent vortices embedded in a turbulent boundary layer. The particles, once suspended, interact with the coherent wake vortices characterizing the rotor flow, and with the finer scale turbulence generated near the ground. The primary objectives are two-flow. First, to gain insight into the particle-vortex dynamics that influence transport near the bed and, second, to advance understanding of the mesoscopic particle velocity field. The latter objective requires very large particle ensembles in order to recover an Eulerian description of the particle field, important to advancing other simulation strategies for two-phase flows. Predictions of the flows for a range of particle and flow parameters will be presented.

Tuesday, November 24, 2009 11:40AM - 1:37PM –
Session PL General Fluid Mechanics II 200A

11:40AM PL.00001 Hydrodynamic coupling of two cylinders in an inviscid fluid , ANDREW TCHIEU, ANTHONY LEONARD, California Institute of Technology, DARREN CROWDY, Imperial College — A general treatment of computing the irrotational flow generated by two objects moving in a planar incompressible and inviscid fluid is studied. We apply the general treatment to a specific case where two circular cylinders are moving in an arbitrary configuration. Explicit equations for the force imposed on the cylinders are derived and are shown to be dependent on the time varying nature of the conformal map used in the problem. The equations can be coupled to the rigid body motion of the body to give a closed system of equations describing their dynamics. We take a closer look into the effects of the forced oscillatory motion of one cylinder on another and the dynamics of coupled near-collision scenarios.

11:53AM PL.00002 Float height of a rectangular plate in planar stagnation flow , PATRICK WEIDMAN, University of Colorado at Boulder — Numerical integrations of the self-similar equations for steady planar fluid motion between infinite stationary parallel plates are reported for the case where the upper plate is impermeable and the lower plate has uniform transpiration. Such a reduction characterized by a single Reynolds number represents a new exact solution of the Navier-Stokes equations. High-Reynolds number asymptotics greatly facilitate numerical integration by a standard shooting technique. The results are applied to model the float height a rectangular plate under gravity when the plate separation distance is small. We use the results to compare the float heights of disks of fixed mass and surface area supported by axisymmetric and planar stagnation flows.

12:06PM PL.00003 Modeling Gust in Model Reduction Framework , MEHDI GHOMMEM, IMRAN AKHTAR, MUHAMMAD HAJJ, ISHWAR PURI, Virginia Tech — Determining gust effects on fluid flows and fluid structure interactions is important in many applications. Direct numerical simulations of these effects may be very expensive. This is especially true in cases where gust parameters could vary. The overall goal of this effort is to develop tools and reduced-order models that are capable of assessing gust effects and that can be used for flow control or uncertainty quantification of flow parameters. As such, we simulate the flow past a circular cylinder with and without an incoming gust. Proper orthogonal decomposition (POD) is performed on the simulated flow data to compute the dominant basis functions (modes) using the method of snapshots. The results show that reduced-order models from steady flow cannot be readily used to develop reduced-order models for the same flow with incoming gust. Methodologies to incorporate gust effect within the reduced-order model are discussed.

12:19PM PL.00004 A Novel Model Reduction Strategy Using Upper Bound Theory[1] , GREG CHINI, University of New Hampshire, CHARLES DOERING, University of Michigan — We propose an original model reduction technique for driven, dissipative infinite-dimensional dynamical systems. Unlike popular – but empirical – POD-based methods, our approach does not require *a priori* data sets from experiments or full PDE simulations and, thus, yields truly predictive reduced models. Instead, the basis functions are computed by solving a constrained, non-local eigenvalue problem drawn from energy stability and upper bound theory. In contrast to *a priori* bases used in spectral expansions, the upper bound eigenfunctions appear to be well suited for the low-order description of strongly driven, spatiotemporally chaotic dynamics, as we demonstrate by applying our methodology to porous medium convection.

[1]GC and CD acknowledge funding from NSF grants DMS-0928098, CAREER OCE-0348981 and DMS-0927587, PHY-0555324, respectively.

12:32PM PL.00005 Laser front distortions due to flow field around a helicopter configuration on hover and forward motion[1] , SIMHA DODBELE, Naval Air Systems Command, Patuxent River, MD — Laser beam front distortions characterizing near field losses of energy from a turret mounted on a generic helicopter have been computed to support the modeling and simulation effort of a directed energy testing. The flow field has been computed using an unstructured computational fluid dynamics method with (a) Reynolds averaged Navier-Stokes method with Spalart-Almaras (SA) turbulence model and (b) time accurate Detached Eddy Simulation with SA turbulence model. The beam distortions have been computed using an aero-optics model developed at the University of Notre Dame. For study (a), the rotor surface is modeled by an actuator disk with a jump in the pressure at the disk plane with the laser passing through the flow field due to the fuselage and the rotor. In the case of study (b), the laser is passing through a time accurate flow field of an isolated rotor blade. The laser front distortions, computed through optical path differences for several azimuth and elevation angles for hover and forward flight conditions, are presented in this study.

[1]Applied Aerodynamics Stores Separation Branch, NAVAIR, Patuxent RIver, MD.

12:45PM PL.00006 Stabilities and transitions of vertical circulating flows induced by a small fan set on a lake surface , TAKASHI NAKAZAWA, HIROSHI SUITO, Okayama Univ., Japan — The purpose for this research is to analyze flow patterns for the swirling upper-lid driven cavity flow in cylindrical coordinates. The swirling upper-lid driven cavity flow is very important for a water quality improvement in lakes. To survey aspects of such flows numerically and mathematically in a simple system, the flows induced by the top boundary condition which forces a horizontal rotating flow is investigated here. Simulations of flows created by the top boundary condition are carried out to obtain steady-state solutions with various Reynolds numbers and analyze stabilities and transitions.

12:58PM PL.00007 Development of the Turbulent Flow in a Bent Pipe , PHILLIP WILSON, University of Canterbury, FRANK SMITH, University College London — The three-dimensional incompressible turbulent flow through a slender bent pipe of simple cross-section is analyzed, the pipe gradually bending the rapid flow through a substantial angle. The ratio of the relative radius of curvature to the magnitude of the turbulent fluctuations is a crucial factor: analysis of the entry region involving exact solutions of the governing equations shows three different downstream developments, depending on the magnitude of that ratio. The main velocity components are found in each case, and one downstream development studied in detail is when turbulence dominates the flow. The present physical situation arises commonly in industrial settings but has been little studied previously. The working applies for any two-tier mixing-length model, and, as a most surprising feature, the fully developed flow far downstream is not unique, being found to depend instead on the global flow behaviour (thus the centre-line velocity is not determined simply by the pressure drop, in contrast to the laminar case).

1:11PM PL.00008 Fast chemical reactions in chaotic flows: predicting the product growth rate[1] , YUE-KIN TSANG, Scripps Institution of Oceanography, UCSD — We consider the fast irreversible bimolecular reaction in a two-dimensional chaotic flow. Simulations show that the reactant concentration decays exponentially with rate λ, and then crosses over to the algebraic law of chemical kinetics in the final stage of the reaction. We estimate the crossover time from the reaction rate constant and the flow parameters. The exponential decay phase of the reaction can be described in terms of an equivalent passive scalar problem, allowing us to predict λ using the theory of passive scalar advection. Depending on the flow configuration, λ is either related to the distribution of the finite-time Lyapunov exponent of the flow, or given in terms of an effective diffusivity. For the former case, we suggest an optimal choice of flow parameters at which λ is maximum.

[1]This work was supported by the National Science Foundation under Grant No. OCE07-26320.

1:24PM PL.00009 Giant Pumpkins , DAVID HU, ALEX ALEXEEV, Georgia Tech — In this combined experimental and theoretical study, we investigate the growth of pumpkins from 1 to 1000 pounds in weight. Time-lapse photography is used to document the growth of pumpkins. Data is presented on the relation between the pumpkins' weights and aspect ratios (height divided by width). We observe pumpkins tend to become squashed (up to 50%) as they increase in size. The lattice-spring method is used to numerically estimate the elasto-plastic forces resisting deformation of the pumpkin. Using levels of plasticity consistent with that of plant cell growth, we find pumpkins shapes consistent with those observed.

Tuesday, November 24, 2009 11:40AM - 1:37PM –
Session PM Acoustics II 200B

11:40AM PM.00001 Investigation of the Near-Field Acoustic Properties of Supersonic Jets with Fluidic Injection using Large-Eddy Simulations , JUNHUI LIU, K. KAILASANATH, RAVI RAMAMURTI, Naval Research Laboratory, DAVID MUNDAY, EPHRAIM GUTMARK, University of Cincinnati — Numerical simulations of Imperfectly Expanded Supersonic Jets from a CD nozzle with fluidic injection have been carried out. A MILES (Monotonically Integrated Large Eddy Simulations) approach with a finite element version of Flux-Corrected Transport algorithm is used. It is found that the fluidic injection alters both shock-cell structures and the near-field noise spectra. It eliminates screech tones and reduces the noise intensities, especially in the region near the main nozzle exit. A wide range of injection conditions, such as the size and number of fluidic nozzles and the injection angle, are simulated and studied. Since both fluidic injection and chevron geometry introduce counter-rotating streamwise vortices that alter the shear layers and the shock-cell structures of the main jet, a detailed comparison of the mechanism of these two approaches for noise reduction are made. Furthermore, the effect of a combination of chevrons and fluidic injection on noise reduction is also investigated numerically. Work sponsored by SERDP.

11:53AM PM.00002 Investigation of the Acoustic Properties of Supersonic Jets with Fluidic Injection on Chevrons: Experimental Results[1] , DAVID MUNDAY, NICK HEEB, University of Cincinnati, EPHRAIM GUTMARK, University fo Cincinnati, JUNHUI LIU, K. KAILASANATH, Naval Research Laboratory — An experimental investigation of two jet noise reduction techniques is presented. These techniques are currently employed on commercial aircraft, and we now apply them to a convergent-divergent nozzle with geometry typical of military aircraft. The acoustic effects of chevrons and fluidic injection on chevrons are quantified by Near-Field and Far-Field acoustic measurements. Experimental tests are shown for overexpanded, underexpanded, and on design nozzle pressure ratios to simulate the entire flight envelope of a military aircraft. In nearly all cases chevrons are shown to reduce noise and eliminate screech tones. Adding fluidic injection to chevrons shows additional far-field noise reduction for underexpanded conditions. This presentation is the experimental portion of a joint numerical/experimental program.

[1]Work sponsored by SERDP.

12:06PM PM.00003 Global Modes of High Speed Jet Noise[1] , JOSEPH W. NICHOLS, SANJIVA K. LELE, PARVIZ MOIN, Center for Turbulence Research, Stanford University, Stanford, CA 94305 — We consider instability wave mechanisms for sound generation in supersonic jets from the perspective of global mode analysis. Using a shift-and-invert Arnoldi method, global modes are extracted from direct numerical simulations of perturbations governed by the fully compressible linearized Navier–Stokes equations. As fully 3D eigenfunctions of the linear stability problem, global modes capture directly effects of base flow non-parallelism. For example, we find a significant sound producing region just downstream of the supersonic core of a transonic jet. In addition, maximum transient growth is computed from an optimal superposition of non-normal global modes. While purely linear, we suggest that this transient growth, composed of several frequencies, may be key to predicting eventual nonlinear mode interactions responsible for low frequency sound production. Finally, sensitivities of the global eigenvalues to base flow modifications are calculated from an overlap of direct and adjoint global modes, suggesting strategies for passive control of jet noise.

[1]This research is supported by NASA contract no. NNX07AC94A.

12:19PM PM.00004 Acoustic resonance in a supersonic ducted jet , VICTOR TOPALIAN, JONATHAN FREUND, University of Illinois at Urbana-Champaign — We consider an overexpanded jet confined by a finite-length duct, which is a model for a particular jet-engine test cell flow in which high-amplitude resonances are observed. This resonance is studied with a two-dimensional model configuration, the flow in which is solved using high-order finite differences on a staggered mesh. A resonant regime was identified for a Mach 1.2 jet, with the amplitude of the acoustic fluctuations of the same order to those observed in test cells. The feedback mechanisms of the resonance will be discussed, and it will be shown that the resonance is suppressed when a damping term designed to remove a small amount of energy from the dominant acoustic mode is added to the equations. The observations from these simulations suggest a scaling that seems to collapse results for the most excited tones observed in several experiments over a broad range of parameters (jet Mach numbers, and duct and nozzle diameters), in both cylindrical and rectangular duct geometries.

12:32PM PM.00005 Flow and noise prediction of transonic turbulent jets including nozzle geometry using LES[1] , MOHAMMAD SHOEYBI, SIMON MENDEZ , PARVIZ MOIN, Stanford University — An unstructured large eddy simulation (LES) method is employed to investigate a turbulent jet in transonic regime. The far-field noise is computed using the integral solution to the Ffowcs Williams-Hawkings equations. The approach has been validated by comparing the near field flow and the far-field sound with the experimental data of Brown and Bridges (AIAA 2006 & 2008) for a jet with a Mach number 0.89 and a temperature ratio 0.84. Although some differences between power-spectra densities from simulation and the corresponding experimental measurements have been observed in regions near the nozzle exit, they are in excellent agreement with experimental data elsewhere. Along the centerline the mean velocity decay is well predicted and turbulent intensity profiles are to within 10-20% of the experimental data. The predicted far-field noise spectra at different polar angles are all within 3dB of the measured experimental ones for Strouhal numbers ranging from 0.05 to 3. Comparisons of flow and sound fields of the heated and unheated jets will be presented.

[1]The work is sponsored by NASA and DOE under the PSAAP program.

12:45PM PM.00006 Far-field radiation of large-scale turbulent structures using wave-packet models , ARNAB SAMANTA, KRISTJAN GUDMUNDSSON, California Institute of Technology, RAMONS REBA, United Technologies Research Center, TIM COLONIUS, California Institute of Technology — Our study concerns sound generation from large-scale turbulent structures of both heated and cold round jets. We obtain predictions for the far-field sound based on wave-packet models that are, in turn, motivated by pressure fluctuations measured experimentally using a microphone array. A Kirchhoff-surface-based projection method has been developed to predict the far-field sound from an equivalent source, defined using the two-point space-time correlations of hydrodynamic pressure measured near a conical surface surrounding the jet plume. The predictions for the aft angles, particularly at lower frequencies are generally good. However, the sensitivity of this projection method to various model and physical parameters is not well understood. Techniques like near-field filtering of the microphone data, which can separate out the acoustic and hydrodynamic components of the pressure signal and also proper orthogonal decomposition (POD), which tends to filter out the uncorrelated fluctuations, are examined to provide better understanding of how these large-scale structures and their wave-packet representations radiate to the far field.

12:58PM PM.00007 Acoustic Measurement of Body Forces Created by Dielectric Barrier Discharge Plasma Actuator and Comparison with Time-Dependent Empirical Model , BEN MERTZ, THOMAS CORKE, SCOTT MORRIS, FLINT THOMAS, University of Notre Dame — Single Dielectric Barrier Discharge (SDBD) plasma actuators have been proven to an effective means of flow control in a variety of applications. As new applications for these actuators emerge, it has become increasingly important to develop a computationally efficient, yet accurate, numercal model to be used in CFD simulations. In previous work, a time-dependent empirical model of a SDBD plasma actuator was developed and validated using time-averaged body force vector results and experimental observations. However, the model is capable of predicting the body force field at the applied a.c. time-scale (on the order of 1 kHz). In order to validate the time-dependent behavior of the model, measurements of the body forces produced by the actuators at the a.c. time-scales are needed. In the present work, we present acoustic measurements of a SDBD plasma actuator in a hemi-anechoic environment. The measurements were taken in 5 degree increments in an arc oriented in the direction of the induced flow. Both the magnitude and phase information were considered and then compared to the results of the lumped circuit element model previously developed.

1:11PM PM.00008 On the reflectivity of sponge regions in compressible flow simulations[1] , ALI MANI, Center for Turbulence Research, Stanford University — In finite-domain compressible flow simulations, one remedy to address lack of boundary information is to gradually relax the flow near the external boundary to a known consistent far-field solution of the Navier-Stokes equations. This treatment, called the sponge treatment, is adopted in many calculations owing to its simplicity, generality and robustness. In practical calculations however, interactions of the sponges with flow features can reflect unphysical signatures into the CFD domain. If the sponge is not carefully designed these reflections can overwhelm the physics of interest particularly when acoustics are concerned. In this work we examine the physics of sponge/flow interactions through analytical and semi-analytical approaches. The reflectivity due to non-linear terms, oblique waves intersecting, and sponge/vortex interactions are separately analyzed. The optimal sponge profiles and the reflection coefficients for asymptotically small or large sponges (compared to flow features) are investigated. These analyses provide estimates of the sponge requirements for CFD calculations in a relatively general framework. In steady state problems the effect of sponge zones on the solution will be discussed.

[1]Supported by AFOSR

1:24PM PM.00009 Application of Stochastic Galerkin Methods (SGM) to Uncertainty Analysis in Computational Aeroacoustics (CAA)[1] , MATTHEW MCCLINTOCK, YASER KHALIGHI, GIANLUCA IACCARINO, Stanford University — In problems of CAA, uncertainties in the flow field data inherent in any turbulent flow calculation, e.g. those due to discretization errors, must be incorporated into predictions of the far-field sound. This is particularly relevant for hybrid CAA methods, which require the flow field data as input and then propagate the effects to the far-field by means of an acoustic analogy. We illustrate the application of a SGM to a canonical problem involving the acoustic far-field generated by a monopole source radiating in the vicinity of a rigid, infinite cylinder in an otherwise quiescent medium, in which the location of the source is assumed to be uncertain. The accuracy and efficiency of this method, in this context, are evaluated by comparison with a local sensitivity analysis and an equivalent Monte-Carlo type simulation, respectively. The present method is transparent to the way in which the flow field data and corresponding uncertainties are obtained, implying that further application to complex flow calculations may be feasible.

[1]Supported by the Department of Energy's PSAAP.

Tuesday, November 24, 2009 11:40AM - 1:24PM –

Session PP Instability: Taylor-Couette 200D

11:40AM PP.00001 Experimental characterization of the Taylor-Couette flow submitted to a radial temperature gradient , ARNAUD PRIGENT, RAPHAËL GUILLERM, INNOCENT MUTABAZI, Le Havre University, KYUNG-SOO YANG, Inha University — We have developed a non-intrusive velocity and temperature fields measurement technique using thermochromic liquid crystals which allows to fully characterize the flow produced in a narrow gap and large aspect ratio Couette-Taylor system submitted to a radial temperature gradient. The aspect ratio and radius ratio of the system are respectively equal to 112 and 0.8. The control parameters are the Grashof number Gr, related to the radial temperature gradient, and the Taylor number Ta, related to the rotation of the inner cylinder. Here, Gr is fixed and Ta is gradually increased. For small values of Ta, the base flow is composed of the circular Couette flow and a vertical flow induced by the radial temperature gradient. Above a critical value of Ta, the destabilization of the base flow gives rise to a spiral pattern. While for small Gr values it corresponds to traveling inclined vortices, for large Gr values it corresponds to a modulated wave-like pattern filling the whole length of the system and rotating at the mean angular velocity of the flow. When Ta is further increased, this wave-like pattern is progressively replaced by a counter-rotating vortices pattern. Numerical simulations of the corresponding Boussinesq-Oberbeck equations provide results in good agreement with experiments.

11:53AM PP.00002 Effects of microbubbles on Taylor-Couette flow , YUJI TASAKA, YUICHI MURAI, TOMOAKI WATAMURA, YASUSHI TAKEDA, Hokkaido University — Effects of micro-bubbles on Taylor-Couette flow has been examined by means of ultrasonic velocity profiling (UVP) for wide range of the Reynolds number, $1 \leq Re/Re_c \leq 18$, where Re_c is the critical Reynolds number for the onset of the primary instability. $O(10\mu m)$-diameter hydrogen bubbles generated by electrolysis of water was dispersed into the fluid layer of water between the vertical, co-axial cylinders. The radius ratio of the cylinders and the aspect ratio are 0.905 and 20, the maximum void fraction estimated by input power for the electrolysis is smaller than 0.1 %. Different flow pattern of the rising bubbles are observed in the spatio-temporal velocity distribution measured by UVP; i.e. free rising and snake-like rising. Axial wavelength of the Taylor vortices show no clear influence of bubbles, but the frequency of the azimuthal traveling wave is reduced by bubbles. Since the number of the traveling waves on the azimuthal plane is the same in the conditions, single phase and with bubbles, the reduction of the frequency means the reduction of the traveling velocity of the wave. Facts to change the traveling velocity, the aspect ratio, the radius ratio, the axial wavelength of the vortices and the number of waves, are no difference on the both cases, therefore, we guess the decrease of the shear rate of the fluid due to micro-bubbles induces this reduction of the traveling speed.

12:06PM PP.00003 Spectral analysis of temporally resolved velocity field data for Taylor-Couette flow , JOHN W. LAAGE, MICHAEL G. OLSEN, R. DENNIS VIGIL, Iowa State University — High speed stereoscopic particle image velocimetry (PIV) data were collected for Taylor-Couette flow for flow regimes varying from wavy through turbulent. The Taylor-Couette apparatus used had an aspect ratio of 34 and a radius ratio of 0.733. The working fluid was an index-of-refraction matched sodium iodide and water solution. Data were obtained for rotational Reynolds numbers ranging from 6 up to 200. Each high speed PIV data set consisted of 2048 images separated in times ranging from 1/60 through 1/2000 second as required by the rotational Reynolds number observed. The time resolved velocity field data were subjected to Fourier Decomposition to find the frequency behavior for a given rotational Reynolds number by calculating the power spectral density. For each investigated rotational Reynolds number, several data sets were averaged together to reduce the effect of observational "noise." Turbulent kinetic energy and Reynolds Stresses were also calculated for the turbulent flow cases. Data at different rotational Reynolds numbers are compared to characterize flow transitions.

12:19PM PP.00004 Characterizing the Subcritical Transition to Turbulence in Taylor-Couette Flow , M.J. BURIN, D.L. DEFOOR, CSU San Marcos — The supercritical transition to turbulence in Taylor-Couette flow is a celebrated paragon of nonlinear dynamics, but the subcritical transition in this system has received much less attention. A few early experiments with the inner cylinder held stationary present to us a suggestive but incomplete picture of a 'catastrophic' transition to turbulence for sufficient outer cylinder speeds. But many questions remain about this shear-driven transition, such as the functional dependence of the critical Reynolds number on rotation, system curvature, cyclonicity, and finite-amplitude perturbations. To address these and related issues a large Taylor-Couette device has recently been constructed, allowing for access to turbulent regimes with either/both cylinders rotating. Some details of this new apparatus along with initial data pertaining to the subcritical transition will be presented.

12:32PM PP.00005 Transient Turbulence in Taylor-Couette Flow: Co/Counter Rotation and Aspect Ratio Effects , DANIEL BORRERO-ECHEVERRY, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology, RANDALL TAGG, Department of Physics, University of Colorado at Denver, MICHAEL SCHATZ, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology — Wall-bounded shear flows typically make the transition to turbulence through a subcritical bifurcation that requires a finite amplitude perturbation. At low Reynolds numbers the lifetime of the turbulent state is finite and increases with increasing Reynolds number. Recent studies have challenged the view that there is a critical Reynolds number above which turbulence becomes sustained. The issue has been further complicated by recent numerical studies that suggest that even if turbulence decays locally, it may become sustained globally if the system is sufficiently large. We address this issue and present lifetime measurements in linearly stable Taylor-Couette flow at various aspect ratios. We also discuss the effects of various boundary conditions and weak counter/co-rotation on the observed lifetimes.

12:45PM PP.00006 Structures in Transitional Taylor-Couette Flows Identified using POD , STAVROULA BALABANI, EBOSHOGWE IMOMOH, JONATHAN DUSTING, Experimental and Computational Laboratory for the Analysis of Turbulence (ECLAT), King's College London, Strand, London WC2R 2LS, UK — The flow in the gap between concentric cylinders, or Taylor-Couette flow, has been used to study transition to turbulence for decades, and is also utilised for various biotechnological and industrial processes. Transitional flow states depend highly on vessel geometry; they are also three-dimensional and often time dependent limiting the use of experimental techniques for their characterisation. In this talk the transition to turbulence in a Taylor-Couette flow is studied by means of time resolved PIV velocity fields and Proper Orthogonal Decomposition (POD). It is found that for the particular geometry studied the transition to turbulence occurs via a quasi periodic regime characterised by a fast moving azimuthal wave (FMAW). Aspects of the FMAW structure, such as a series of co-rotating vortices that increase in strength away from the endwalls, are also revealed by spatially resolved POD.

12:58PM PP.00007 Controlling Transition in Taylor-Couette Flow with Spatial Forcing , YASSER ABOELKASSEM, ANNE STAPLES, Virginia Polytechnic Institute and State University — The linear stability of the flow in the (narrow) annular gap between two infinitely long cylinders, driven by an axisymmetric sinusoidal perturbation to the radius of the inner cylinder in the axial direction is analyzed. A closed-form solution for the basic flow in the system is derived. Experiments and computational investigations of this system have given differing results. In the seminal experiment performed by Ikeda and Maxworthy (Phys. Rev. E, 1994), the perturbation was found to have no effect on the first stability boundary. In subsequent theoretical investigations, authors have concluded that circular flow cannot exist in the modified system, and that the basic flow is Taylor Vortex Flow. In this study, we find that while the perturbation seems to always be destabilizing, circular flow does indeed exist in the system, in agreement with experimental observations. For small to moderate forcing amplitudes, the critical Taylor number for the first transition is only reduced slightly, by an amount that depends on the forcing amplitude and wavelength. The reduction in the first critical Taylor number is speculated to lie within the margin of error in the experiments performed by Ikeda and Maxworthy.

1:11PM PP.00008 Pattern formation in plane Couette flow turbulence , YOHANN DUGUET, PHILIPP SCHLATTER, DAN S. HENNINGSON, KTH Mechanics, Stockholm, Sweden — Plane Couette flow is the flow between two counter-sliding plates of velocity U separated by a gap $2h$. Despite the linear stability of the laminar base flow, sustained turbulence is observed experimentally above $Re \sim 300$, where $Re = \frac{Uh}{\nu}$ (ν is the kinematic viscosity of the fluid). Whereas featureless turbulence is seen for $Re \geq 400$, lower-Re experiments have shown the appearance of turbulent stripes, inclined with respect to the base flow, interspersed with quiescent, nearly laminar regions. Direct numerical simulation using a spectral code in an unusually large computational domain ($800h$ in length and $356h$ in width with periodic boundary conditions) is performed here to highlight the large-scale self-organisation of the flow, out-of-reach in former simulations. The system evolves towards a fragmented large-scale pattern, where several competing inclinations can coexist. We suggest a new way to study the angle selection of those turbulent patterns, based on the computation of edge states in a smaller computational domain of size $80h \times 80h$. We can show that angles close to 40° are prefentially chosen by the system close to the threshold and that this range of angles increases with the amplitude ofthe initial perturbation. This demonstrates that the pattern selection is linked to the subcritical nature of the transition process.

Tuesday, November 24, 2009 11:40AM - 1:50PM –

Session PQ Industrial Applications 200E

11:40AM PQ.00001 Study of the turbulent wake behind a tidal turbine through different numerical models[1] , AMIR TEYMOUR JAVAHERCHI MOZAFARI, ALBERTO ALISEDA, SYLVAIN ANTHEAUME, JOSEPH SEYDEL, BRIAN POLAGYE, University of Washington — As developing sources of renewable energy becomes a critical priority, research in this field become more essential. A novel method to produce clean renewable energy is extraction from ocean tides via a turbine. Although energy generation from tidal currents has many similarities to wind, the balance between kinetic and potential energy is a key element in tidal channels that invalidates "Betz's" limit. Other practical differences arise from the concentrated nature of tidal resources which impose very close turbine spacing for economic reasons. These, together with the potential influence of geometric constraints imposed by free surface and tidal channel walls, makes the study of the turbulent wake in tidal energy extraction a very important problem in development of this technology from economical and environmental aspects. We will present numerical simulations of turbulent wake behind a well characterized two-bladed turbine using a hierarchy of different models: Actuator Disk, Virtual Blade, the Single and Multiple Reference Frame and Sliding Mesh model with various boundary conditions and inlet velocity profiles. We will compare the results, discuss the differences among these models and the potential for each one to answer questions about optimization of energy extraction and environmental impacts.

[1]Supported by DOE through the National Northwest Marine Renewable Energy Center.

11:53AM PQ.00002 Wind energy conversion affected by turbulent wind conditions , JOACHIM PEINKE, MATTHIAS WAECHTER, PATRICK MILAN, ForWind University of Oldenburg — Wind turbines are put up into the turbulent boundary layer of windy regions. The standard operational description is mainly based on simple statistical quantities like averaged wind speeds and turbulence degree (standrad deviation). Short time fluctuations are expected to be Gaussian. In this contribution we show that the intermittent - non Gaussian structure of small scale wind turbulence plays an important role for the working condition of a wind turbine and leads to additional mechanical loads as well as to fluctuations of the power production with many short time extreme events.

12:06PM PQ.00003 Flow structure and induction factor of a lightly loaded wind turbine model in a wind-turbine array boundary layer[1] , JOSE LEBRON, Rensselaer Polytechnic Institute, RAUL BAYOAN CAL, Portland State University, HYUNG SUK KANG, The Johns Hopkins University, LUCIANO CASTILLO, Rensselaer Polytechnic Institute, CHARLES MENEVEAU, The Johns Hopkins University — Wind-tunnel experiments are carried out in order to study the structure of the flow within a 3 by 3 array of lightly loaded wind turbine models operating inside of a boundary layer. Particle-Image-Velocimetry measurements are performed in a volume surrounding a wind turbine model located on the center of the last row. Data gathered is used to compute time-averaged mean velocity and turbulence quantities in 18 planes surrounding the wind turbine model, and missing data are obtained through bilinear interpolation in space. The induction factor is estimated by computing the volumetric flux and mean velocities through circles centered at hub-height both upstream and downstream of the wind turbine model. Computation and visualization of the streamtube is performed by tracking the evolution of (virtual) fluid particles that pass through the (interpolated) rotor disk location. The effects of wall blockage, mean velocity shear, and turbulence stresses and mixing on the streamtube geometry are considered.

[1]Supported by the NSF (CBET-0730922) and NSF (DGE-0742436).

12:19PM PQ.00004 Laboratory model of the airborne jet-ski train riding on a soft porous track[1] , PARISA MIRBOD, YIANNIS ANDREOPOULOS, SHELDON WEINBAUM, The City College of the City University of New York, New York, NY 10031, USA — A small laboratory scale prototype of the airborne jet train and its track has been designed and constructed to measure the pressure signals as the planfrom passes over a particular location. We considered a lift weight W of 1 kg fully supported when airborne by a planform that is 65 cm long and 10 cm wide symmetrically positioned beneath the prototype. Our prototype model used the same porous media, described in Mirbod et al. (2009) J. Fluid Mech. 619:127 for the full scale AJT, where the Darcy permeability Kp, is 3.4×10^{-9} m^2. We have performed a parametric study to estimate the primary lift-off velocity U as a function of compression ratio, k for this value of Kp and W. We considered the simpler case where both the variation of Kp with compression and the small lift force due to the fiber phase are neglected in predicting the pressure distribution on the planform. One then integrated this pressure distribution over the surface of the planform and sets F in the expression for the dimensionless force $F = F\,Kp(H)/\mu L2Uw$ equal to the weight W of the prototype model. This solution, which provided an expression for U as a function of k, is used to predict primary lift off velocity for a laboratory model and to obtain an initial estimate for the pressure distribution beneath the planform.

[1]This work is supported by NSF Grant CTS #0432229.

12:32PM PQ.00005 Stress transfer through fibrous materials in wicking experiments[1] , DARIA MONAENKOVA, TARAS ANDRUKH, KONSTANTIN KORNEV, Clemson University — Due to the recent progress in preparation of fibers and nanofibers with different properties, the idea of smart textiles attracts much attention. In many situations the probes and sensors are designed for bio fluid detection. The liquid penetration in fibrous materials causes their deformations including stretching, twisting, wrinkling, buckling etc. The most of researches on wicking properties of textiles are focused on determination of media permeability and ignore the specific features of fibrous materials. On the other hand the theoretical works on quantitative analysis of the deformation effects in porous materials filled with liquids are mostly focused on deformation of fully saturated samples. The fundamental understanding of the stress transfer through the fiber network is crucial for sensors development, but to the best of our knowledge, the stress analysis in the fibrous materials absorbing liquids has never been discussed in the literature. This paper sets a physical basis for analysis of absorption processes in nanotubular and nanofibrous materials. We study absorption of droplets by yarns and webs made of fibers, develop a theory which explains the stress distribution in fibrous materials and checked this theory on wicking experiments. The reported theory and experiments propose a new area of research on absorption-induced deformations of fibrous materials.

[1]We acknowledge support from NSF, grant number CMMI-0826067 and NTC, project M08-CL10.

12:45PM PQ.00006 A flow intensification model for granular filter applications , ZHAOHUI QIN, RICHARD PLETCHER, RODNEY FOX, SHANKAR SUBRAMANIAM, Iowa State University — a flow intensification model is proposed. We first give an estimation of the flow intensification factor, and then the velocity field in the vicinity of a single granule subject to the intensified flow is obtained. Creeping flow is assumed and Happel's model is used to represent the granular media. Based on the flow field, the initial collector efficiency η_0 is calculated from trajectory analysis and compared with experimental data. The reasonably good agreement between the theory and experiments suggests that the current model might be employed as a starting framework for further theoretical development and numerical modeling for granular filter applications.

12:58PM PQ.00007 Swirling jet nozzle design for seabed excavation[1] , J. ORTEGA-CASANOVA, N. CAMPOS, R. FERNANDEZ-FERIA, University of Malaga (Spain) — We have investigated experimentally the seabed excavation performance of several swirling jets, generated by swirl vanes with adjustable angles and different geometries inside a nozzle, impinging against a sand bed for several Reynolds numbers and different impinging distances. The velocity profiles of the swirling jets at the nozzle exit have been measured using LDA, for the different nozzle configurations and Reynolds numbers, and the main features of the generated footprints on the sand bed have been measured by image processing of photographs of the bed illuminated by a laser sheet. Numerical simulations of the flow and visualizations techniques have been used to understand the excavation characteristics of the different swirling jets. It is found that the maximum excavation power is produced, at moderate impinging distances, by jets with a maximum swirl intensity in an annular region surrounding a central core with almost vanishing swirl, and a marked maximum of the axial velocity at the axis. This particular velocity profile of the swirling jet is generated by one of the nozzle configurations considered when the Reynolds number is above a threshold value.

[1]Supported by the Ministerio de Ciencia e Innovación (Spain) Grant no. FIS07-60161. We acknowledge J.H. Redding (SILT Co., UK) for sharing with us his insight on the subject.

1:11PM PQ.00008 Internal Combustion Engine Flows[1] , AMY MCCLENEY, Embry-Riddle Aeronautical University, PAUL PUZINAUSKAS, KENDRICK GIBSON, University of Alabama — An automobile engine's performance can be enhanced by a more complete combustion reaction which results in less fuel consumption and lower emissions. The combustion improvement can be accomplished through an increase in turbulence from tumble flow, a circulatory motion inside the cylinder. In previous research, this increase is created by changing the intake ports on the engine so the flow is more precisely directed in the cylinder. In this study, the following three experiments were conducted: a detailed vane characterization experiment to direct the flow; a study of the effect of the piston shape on the flow; and a seeding settling experiment to determine experimentation quality. These experiments offer insight into the flow structure inside the test cylinder, as observed using particle image velocimetry and impulse swirl meter testing.

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

1:24PM PQ.00009 Visualization of a cryogenic jet simulating leak from a liquid hydrogen storage tank[1] , TIM ROSE, AJAY AGRAWAL, University of Alabama — Hydrogen is considered an alternative fuel in propulsion and power generation due to fuel economy standards and environmental pollution. However, if an accidental leak were to occur in a hydrogen storage tank, the discharged fuel could find an ignition source and produce an explosion. A barrier wall can be used to contain the leak from the storage tank, therefore protecting equipment and people from the explosion. Past studies have investigated the jet/barrier wall interaction, in a laboratory setting, with fuel stored as a gas. Hydrogen fuel stored as a liquid offers higher energy density. In this work, we have studied the leak at cryogenic conditions due to liquid storage parameters. Jet fluid structure is visualized in a laboratory setting using helium as the supersonic jet fluid. High-speed rainbow schlieren deflectometry (RSD) images are used to show instantaneous flow structure of jet (leakage point) and barrier wall interactions. Results show the jet inlet temperature leads to significant differences in the spread angle and the extent of fuel-air mixture region adjacent to the barrier wall.

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

1:37PM PQ.00010 Preconditioners for incompressible flows , CORNELIS VUIK, MEHFOOZ UR REHMAN, GUUS SEGAL, Delft University of Technology — We consider solution methods for large systems of linear equations that arise from the finite element discretization of the incompressible Navier-Stokes equations. These systems are of the so-called saddle point type, which means that there is a large block of zeros on the main diagonal. To solve these type of systems efficiently, several block preconditioners have been published. We propose a new block preconditioner MSIMPLER which is a modified version of the SIMPLER solver proposed by Patankar. We compare the performance of the block preconditioners: pressure convection diffusion (PCD), least squares commutator (LSC), and augmented Lagrangian based (AL) preconditioners with the MSIMPLER preconditioner. These preconditioners are first compared for academic problems generated by the IFISS package. Thereafter, we compare the best preconditioners for industrial problems using the SEPRAN FEM package. It appears that the MSIMPLER preconditioner is in general the best method. Finally we also consider the solution of the Stokes problem with variable viscosity. We consider smoothly varying viscosity and jumps in the viscosity. It appears that a special preconditioner is independent of the grid-size and independent of the viscosity variation. Applications from geophysics are used to illustrate the performance.

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PR Convection III 200F

11:40AM PR.00001 A novel way to measure molecular diffusivity[1] , THOMAS PEACOCK, MICHAEL ALLSHOUSE, MIT — We present a new technique for measuring molecular diffusivity. The method exploits the phenomenon of diffusion-driven flow, in which buoyancy-driven convection occurs on sloping surfaces due to the diffusive-flux boundary condition affecting the local fluid density. By measuring the volume flux drawn into the boundary layer flow on a sloping surface, and exploiting the properties of an analytical solution, the molecular diffusivity is determined. The results are compared with predictions from numerical simulations. This approach allows one to readily investigate such issues as the effect of concentration on molecular diffusivity.

[1]This work is supported by the NSF.

11:53AM PR.00002 Sailing On Diffusion , MICHAEL ALLSHOUSE, MIT, MIKE BARAD, Stanford, THOMAS PEACOCK, MIT — When a density-stratified fluid encounters a sloping boundary, diffusion alters the fluid density adjacent to the boundary, producing spontaneous along-slope flow. Since stratified fluids are ubiquitous in nature, this phenomenon plays a vital role in environmental transport processes, including salt transport in rock fissures and ocean-boundary mixing. Here we show that diffusion-driven flow can be harnessed as a remarkable means of propulsion, acting as a diffusion-engine that extracts energy from microscale diffusive processes to propel macroscale objects. Like a sailboat tacking into the wind, forward motion results from fluid flow around an object, creating a region of low pressure at the front relative to the rear. In this case, however, the flow is driven by molecular diffusion and the pressure variations arise due to the resulting small changes in the fluid density. This mechanism has implications for a number of important systems, including environmental and biological transport processes at locations of strong stratification, such as pycnoclines in oceans and lakes. There is also a strong connection with other prevalent buoyancy-driven flows, such as valley and glacier winds, significantly broadening the scope of these results and opening up a new avenue for propulsion research.

12:06PM PR.00003 Diffusive or convective regime: role of vibrations in Space experiments , VALENTINA SHEVTSOVA, MRC, University of Brussels — Despite the dramatic decrease of gravity level in spacecrafts, diffusive transport in liquid phase can be significantly perturbed by convection. Although it is recognized that on-board g-jitters may have a major impact on diffusion and thermal diffusion measurements, very few experiments have been carried out in the past. There is a lack of experimental data to validate numerous numerical studies. In the frame of ESA program the experiment SODI / IVIDIL (Influence of Vibration on Diffusion in Liquids) will be performed in September-October 2009 on ISS. Purpose of the project is to measure thermal and isothermal diffusion coefficients in binary systems subjected to controlled vibrations under different values of amplitude and frequency. The measurements will be repeated at wide range of amplitudes and frequency in order to validate the results. The IVIDIL experiment will be performed in two cell arrays each composed of two cells: the primary cell that is probed by MZI and the companion cell that is filled with tracer particles and is probed by digital holography for PIV. Different cell arrays are filled with various liquids. Over the experiment run, the cells are temperature controlled. IVIDIL science team foresees to present first experimental results.

12:19PM PR.00004 Optimal Prandtl number for heat transfer in rotating Rayleigh-Bénard convection[1] , RICHARD STEVENS, Twente University, HERMAN CLERCX, Eindhoven University, DETLEF LOHSE, Twente University — The heat transfer in Rayleigh-Bénard convection (RBC) is determined by the Rayleigh number Ra and the Prandtl number Pr [2]. In case of rotation about the vertical axis the third dimensionless control parameter is the Rossby number Ro. Here we present numerical data for the heat transfer in rotating RBC for $Ra = 10^8$ as a function of Pr and Ro. When Ro is fixed the heat transfer enhancement with respect to the non- rotating value as function of Pr shows a maximum. This maximum is due to the reduced efficiency of Ekman pumping when Pr becomes too small or too large. When Pr becomes too small the heat that is carried by the vertical vortices spreads out in the middle of the cell, i.e. it makes Ekman pumping less efficient, due to the larger thermal diffusivity [3]. For higher Pr the thermal boundary layers (BLs) are much thinner than the kinetic BLs and therefore the Ekman vortices do not reach the thermal BL. This means that the fluid that is sucked into the vertical vortices is colder than for lower Pr and this limits the efficiency of Ekman pumping at high Pr.

[1]Stichting Fundamenteel Onderzoek der Materie (FOM)

[2]Ahlers et al. *Rev. Mod. Phys.* **81**, 503 (2009)

[3]Zhong et al. *Phys. Rev. Lett.* **102**, 044502 (2009); Stevens et al. *Phys. Rev. Lett.* **103**, 024503 (2009)

12:32PM PR.00005 Convection-driven pattern formation in lawn grasses , SALLY THOMPSON, Nicholas School of The Environment, Duke University, KAREN DANIELS, Dept. of Physics, NC State University — Spatial patterns of 'dead' lawn grass have often been ascribed to Turing-type reaction-diffusion processes related to water scarcity. We present an alternative hypothesis: that the air within the grass canopy is unstable to a convective instability, such that chill damage caused by falling cold air is responsible for the creation of brown and green bands of grass. This hypothesis is consistent with several features of small-scale vegetation patterns, including their length scale, rapid onset and transient nature. We find that the predictions of a porous medium convection model based are consistent with measurements made for a particular instance of lawn-patterning in North Carolina.

12:45PM PR.00006 An Experimental Study of Flow Reversals in Quasi-2D Rayleigh-Benard Convection[1] , RUI NI, TAK-SHING CHAN, SHENG-QI ZHOU, HENG-DONG XI, Department of Physics, The Chinese University of Hong Kong, Shatin, Hong Kong, China, KE-QING XIA, Department of Physics, The Chinese University of Hong Kong, Hong Kong, KAZU SUGIYAMA, ENRICO CALZAVARINI, CHAO SUN, DETLEF LOHSE, Department of Science and Technology, University of Twente, The Netherlands, SIEGFRIED GROSSMANN, Fachbereich Physik, Philipps-Universität Marburg, Germany — We present an experimental study of flow reversals of the large-scale circulation in turbulent Rayleigh-Benard convection in a quasi-2D rectangular convection cell with aspect ratio 0.84. Using water as the convecting fluid the range of Rayleigh number varied from around 10^8 to 10^{10} and the Prandtl number is around 5.8 in the experiment. From local temperature measurements it is found that, contrary to the 3D case, the rate of flow reversals has a strong dependence on the Rayleigh number, i.e. it decreases with increasing Ra. By using thermal plumes as tracer particles, we use shadowgraphs to obtain the velocity field of plume clusters. The results suggest the role played by the plumes in the reversal process.

[1]Work supported by the Research Grants Council of Hong Kong SAR (Project Nos. CUHK 403807).

12:58PM PR.00007 Thermal and PIV measurements of heat convection in a tilted channel , JEAN-CHRISTOPHE TISSERAND, MATHIEU CREYSSELS, BERNARD CASTAING, FRANCESCA CHILLA, ENS Lyon — The Rayleigh-Benard system, in which a fluid is cooled from above and heated from below, is one of the most studied systems in thermal convection. Nevertheless, in this configuration, the neighborhood of the plates controls the heat transfer. Subsequent, we have built a system (a vertical channel) where the flow forgets the cold and the hot plate. Moreover, we have built a structure which allows to tilt this channel from an angle of 0 to 90 degrees. The main goal of this paper is to highlight how the flow in the bulk of the inclined channel is. In a first part, we present new measurements obtained thanks to particle image velocimetry (PIV) technique and a model which interprets our results. The second part of the paper will be focused on thermal measurements and on the influence of the difference of temperature between the two plates.

1:11PM PR.00008 Buoyancy-Induced Columnar Vortices , MARK SIMPSON, ARI GLEZER, Georgia Institute of Technology — Large-scale inherent instability of a thermally stratified air layer is exploited for deliberate formation of intense vertical column vortices. In hot-climate regions, 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. Optical diagnostics includes high-speed video imaging and particle image velocimetry. It is shown that vortices having a nominal 10 cm diameter core with nearly-uniform vorticity distribution can be triggered by and "anchored" to small ground protrusions, and their circulation and angular momentum can be controlled by geometrical modifications of these surface protrusion and by simple flow vanes.

1:24PM PR.00009 Convective instability of a fluid layer induced by an inclined temperature gradient , ALEJANDRO SEBASTIAN ORTIZ-PEREZ, LUIS ANTONIO DAVALOS-OROZCO, Departamento de Polímeros, Instituto de Investigaciones en Materiales, Universidad Nacional Autónoma de México — We present new results of the convective stability of a shallow layer of Newtonian fluid under the presence of an inclined temperature gradient. The horizontal component produces the so called Hadley circulation. The vertical component induces the well known Rayleigh convection with vertical Rayleigh number Ra_v. Here, the numerical results of Nield (1994) are corrected and extended for larger horizontal Rayleigh numbers, Ra_H. Two modes, the longitudinal and the transversal, compete as the most unstable in different regions of the parameters space which also includes the Prandtl number Pr. The calculations of these corrections were motivated by the paper by Kaloni and Qiao (1996) where they present only particular cases of the nonlinear energy stability of the problem. The horizontal temperature gradient stabilizes until certain magnitude after which the curves of criticality start to decrease because the magnitude of Ra_H is very near to its critical value. At this critical value the vertical Rayleigh number is zero. Our results agree very well with those of Kuo and Korpela (1986) (presented also in Wang and Korpela (1989)) who made numerical analysis of the linear instability of the Hadley circulation, presenting corrections to the classical paper by Hart (1972).

1:37PM PR.00010 New results in bioconvective linear stability of gravitactic microorganisms , ILDEBRANDO PEREZ-REYES, LUIS ANTONIO DAVALOS-OROZCO, Instituto de Investigaciones en Materiales, Departamento de Polimeros, Universidad Nacional Autonoma de Mexico — New results on the linear bioconvective instability of a suspension of gravitactic microorganisms have been calculated. Use is made of the model equations presented by Ghorai (2000,2007) some years ago but that have not been used to determine the linear instability in an infinite horizontal fluid layer. The hydrodynamic stability is characterized by dimensionless parameters such as the bioconvection Rayleigh number Ra, the gyrotaxis number G, the motility of microorganisms d and the wavenumber k of the perturbation. Analytical and numerical solutions are calculated. Two numerical methods are used for the sake of comparison. They are a shooting method and a Galerkin method. Marginal curves of Ra against k for fixed values of d and G are presented along with curves corresponding to the variation of the critical values of Rac and kc. The important result of this work is that those critical values are compared with the experimental data presented in Table I of Bees and Hill (1997) and Table II of Bees and Hill (1998), where the gyrotactic algae Chlamydomonas nivalis is used, and it is found that the agreement is very good. A discussion of the results will be given.

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PS Geophysical: General II - Stratified Fluids and Turbulence 200G

11:40AM PS.00001 Energetics of vertical fluid particle dispersion in stably stratified turbulence , SEUNGBUM JO, KEIKO NOMURA, JAMES ROTTMAN, University of California, San Diego — The vertical dispersion of fluid particles in stably stratified turbulence is investigated. We present an analysis framework which describes the associated flow energetics in the Lagrangian frame. The total potential energy of a marked fluid particle is considered in terms of the available potential energy, associated with the nonequilibrium displacement, and the equilibrium (minimum) potential energy, associated with the change in particle equilibrium height. The corresponding evolution equations elucidate the key sequence of processes and clarify previous interpretations of the transport mechanisms. The analysis shows that in the case of stationary flow, the rate of mean square displacement is equal to the rate of mean square equilibrium displacement which is given by the scalar dissipation rate. The analysis is demonstrated using direct numerical simulations of stationary sheared turbulence. A dispersion model is developed and compared with previous models.

11:53AM PS.00002 On spectral energy transfer in strongly stratified flows[1] , JAMES RILEY, VISHAL VASAN, OSCAR FLORES, University of Washington, P.K. YEUNG, Georgia Institute of Technology — The fluid dynamics of many regions of the atmosphere and oceans are characterized by being strongly affected by stable density stratification, but weakly affected by the earth's rotation. This is typical of the open ocean, e.g., on horizontal scales from a few meters up to hundreds of meters. One approach to understanding and predicting such flows is by examining their nonlinear, spectral energy transfer. At horizontal scales strongly affected by stratification, vertical motion as well as the stretching of vertical vorticity is suppressed. In addition the stable stratification supports the propagation of internal waves. These features impose strong constraints on spectral energy transfer. We will present the results of direct numerical simulations of strongly stratified flows with Taylor-Green initialization. The simulations are performed using highly scalable codes run on massively parallel computers; the computational domains are highly anisotropic with up to 2048 X 2048 X 256 grid points. This allows sufficient resolution to examine both the downscale as well as the upscale transfer of energy, both which are found to be dynamically significant.

[1]NSF Grant No. OCI-0749209

12:06PM PS.00003 Energy spectra of stably stratified turbulence , YOSHIFUMI KIMURA, Nagoya Univ., JACKSON HERRING, NCAR — Energy spectra for forced stably stratified turbulence are investigated numerically using the Direct Numerical Simulations (DNS) with 1024^3 grid points. 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 divided into the vortex mode (ϕ_1) and the wave mode (ϕ_2). The ϕ_1 and ϕ_2 spectra as a function of horizontal wave numbers, $k_\perp$, has the form of

$$E_{\perp\Phi_1}(k_\perp) = \begin{cases} \alpha\eta_{\perp\Phi_1}^{2/3} k_\perp^{-3} & (k_\perp < k_c) \\ C_K \varepsilon_{\perp\Phi_1}^{2/3} k_\perp^{-5/3} & (k_\perp > k_c) \end{cases},$$

$$E_{\perp\Phi_2}(k_\perp) = \begin{cases} \beta\sqrt{N\varepsilon_{\perp\Phi_2}} k_\perp^{-2} & (k_\perp < k_c) \\ C_K \varepsilon_{\perp\phi_2}^{2/3} k_\perp^{-5/3} & (k_\perp > k_c) \end{cases}.$$

where $\eta_{\perp\phi_1}$ and $\varepsilon_{\perp\phi_2}$ are the horizontal enstrophy dissipation based on the ϕ_1 energy and the horizontal energy dissipation based on the ϕ_2 energy, respectively. For both cases, $C_K \approx 1.2 \sim 2.0$ is obtained being close to the Kolmogorov constant. To understand the reason for the steeper spectra than the Kolmogorov -5/3 for large scales, inviscid calculations (truncated Euler's equation) without forcing are conducted. We verified that emergence of steeper spectra for large scales and thermalization spectra for small scales.

12:19PM PS.00004 On the turbulent Prandtl number in homogeneous stably stratified turbulence , SUBHAS VENAYAGAMOORTHY, Colorado State University, DEREK STRETCH, University of KwaZulu-Natal — We derive a general relationship for the turbulent Prandtl number Pr_t for homogeneous stably stratified turbulence from the turbulent kinetic energy and scalar variance equations. A formulation for the turbulent Prandtl number Pr_t is developed in terms of a mixing lengthscale L_M and an overturning lengthscale L_E, the ratio of the mechanical to scalar time scales $T_L/T_\rho = (k/\epsilon)/(\frac{1}{2}\overline{\rho'^2})/\epsilon_\rho)$ and the gradient Richardson number Ri. We show that our formulation for Pr_t is appropriate even for nonstationary (developing) stratified flows since it does not include the reversible contributions in both the kinetic energy production and buoyancy fluxes that drive the time variations in the flow. Our analysis of direct numerical simulation data of homogeneous sheared turbulence shows that the ratio $L_M/L_E \approx 1$ for weakly stratified flows. We show that in the limit of zero stratification, the turbulent Prandtl number is equal to the inverse of the ratio of the mechanical to scalar time scales, T_L/T_ρ. We propose a new parameterization for Pr_t in terms of the gradient Richardson number Ri and use data from stably stratified direct numerical simulations to support it. The formulation presented here provides a general framework for calculating Pr_t that will be useful for turbulence closure schemes in numerical models.

12:32PM PS.00005 Double-diffusive convection in narrow-aspect cylinders – experimental data and CFD simulations[1] , STEPHEN WEBB, Sandia National Laboratories, RENETA DIMITROVA, SUHAS POL, H.J.S. FERNANDO, Arizona State University — An experimental and numerical investigation has been conducted on the evolution of double-diffusive convection in a narrow-aspect cylinder in the diffusive regime in which the cylinder is subjected to bottom and side-wall heating. The laboratory experiments included filling of a narrow aspect ratio tank with linearly stratified salt-water solution. This fluid of known stratification was heated from the side and the bottom to form multiple mixed layers of fluid separated by diffusive interfaces. A micro conductivity and temperature probe was used to obtain the measurements of salinity and temperature and to discern the location of the interfaces. These results are compared to the case of bottom heating only. The numerical simulations have been performed with the FLUENT computational fluid dynamics (CFD) code. Laminar and turbulent simulations have been conducted for one of the experiments. Comparison of laboratory and numerical results shows reasonable agreement.

[1]Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company for the United States Department of Energy National Nuclear Security Administration under contract DE-AC04-94AL85000.

12:45PM PS.00006 Parametric study of the spectral characteristics of the internal gravity wave field emitted by a stratified turbulent wake with non-zero net momentum[1] , PETER DIAMESSIS, AMMAR ABDILGHANIE, Cornell University — The internal wave field emitted by the turbulent wake of a towed sphere in a linearly stratified fluid is studied numerically for a range of sphere-based Reynolds and Froude numbers. The full three-dimensional time-dependent governing equations are solved using a parallel spectral multidomain penalty solver. Hovmöller (space-time) diagrams of the horizontal divergence field are constructed in the vertical and lateral directions. Spectral analysis of these diagrams enables the identification of the frequency and wave-number of the most energetic waves. Ensemble averaging at multiple stations in the stream-wise direction is performed to remove the influence of streamwise variability. In agreement with previous studies of wave emission from forced turbulent mixed regions, the spectrum exhibits pronounced peaks in a narrow range of frequencies (and hence angles of propagation) and wave numbers. Finally, the role of intermediate-time secondary instabilities and turbulence inside the wake core on wave radiation is discussed.

[1]Funded by the Office of Naval Research.

12:58PM PS.00007 Wind-driven turbulent oscillating channel flow under a stable stratification , WERNER KRAMER, HERMAN CLERCX, Eindhoven University of Technology, VINCENO ARMENIO, Università degli Studi di Trieste — We use large-eddy simulations of a turbulent oscillating channel flow to investigate the effects of wind forcing and stratification. This kind of flow is a model for the tidal driven flows in estuaries. In our studies the wind is aligned with the oscillating tidal flow, leading to a pulsating mean flow. Turbulent fluctuations are enhanced in the shear layers present at the no-slip bottom and below the free-surface boundary. A stable density stratification arises from a constant solar heating of the free surface. We will report on the mixing properties and on the structure of the turbulent flucations of such flows. During phases with high turbulent activity the whole fluid column is well-mixed except for the free-surface layer. When turbulence levels drop the stratification extends over the entire fluid column, but does not penetrate the bottom boundary layer. A stable density stratification suppresses vertical transport. In the unstratified case turbulence generated in top and bottom layer interact. Whether the interaction can be suppressed by a stable density stratification is an interesting point for further investigation.

1:11PM PS.00008 Internal waves in nonuniform stratifications[1] , MANIKANDAN MATHUR, BRIAN DOYLE, THOMAS PEACOCK, Department of Mechanical Engg., Massachusetts Institute of Technology — Internal waves are propagating disturbances in a stably stratified fluid. Linear internal wave propagation in a stratification with constant Brunt-Vaisala frequency N is well-understood, both theoretically and experimentally. When N varies with the vertical coordinate z, however, propagating internal waves bend with respect to the vertical, and can get scattered in a non-trivial manner, depending on the exact profile $N(z)$. Here, we discuss two scenarios, (i) a finite-thickness N_2-layer sitting atop a semi-infinite N_1-layer, and (ii) a finite-thickness N_2-layer sandwiched between two semi-infinite N_1-layers. The former is shown to support wave beam ducting even in the absence of evanescent layers, and is used to explain the observations of a vanishing wave beam near the upper ocean at the Keana ridge, Hawaii. The latter scenario, in the inviscid limit, is shown to be equivalent to the classical multiple beam light interferometer, and hence results in selective wave transmission based on spatial wavelengths. Results from laboratory experiments and linear viscous theory are presented for both the cases.

[1]This work is partially supported by the ONR grant N00014-09-0282.

1:24PM PS.00009 Slow oscillation of passive scalar fluxes in stratified turbulence , HIDESHI HANAZAKI, TOHRU OKAMURA, Kyoto University — Scalar fluxes in decaying stratified turbulence are investigated when vertical mean gradients of density and passive scalar coexist. Direct numerical simulations show temporal oscillations of a passive scalar flux which contain a component of double-period oscillation, as predicted by the rapid distortion theory. The slow modes appear always when there are initial density fluctuations. For passive scalars with high Schmidt number (Sc>2Pr/(1+Pr), Pr: Prandtl number of the active scalar), the slow modes become dominant at large times, and the turbulent diffusion coefficient of a passive scalar Kc decays more slowly than the turbulent density diffusion coefficient. On the other hand, at low Schmidt numbers (Sc<2Pr/(1+Pr)<2), slow mode suffers strong initial decay, so that Kc and Krho decay at similar rates. These results illustrate the importance of molecular diffusivity, initial conditions and unsteadiness in strongly stratified turbulence. Applicability of RDT is tested for a range of Reynolds numbers and Froude numbers, showing strong sensitivity to the Froude number but with weak sensitivity to the Reynolds number, in agreement with the scaling analysis.

1:37PM PS.00010 Tangential oscillations of a circular disk in a viscous stratified fluid , STEFAN LLEWELLYN SMITH, Department of Mechanical and Aerospace Engineering and Scripps Institution of Oceanography, UCSD, TONY DAVIS, Department of Mechanical and Aerospace Engineering, UCSD — A complete solution is obtained for the wave field generated by the time-harmonic edgewise oscillations of a horizontal circular disk in an incompressible stratified viscous fluid. The linearized equations of viscous internal waves and the no-slip condition on the rigid disk are used to derive sets of dual integral equations for the fluid velocity and vorticity. The dual integral equations are solved by analytic reduction to sets of linear algebraic equations. Asymptotic results confirm that this edgewise motion no longer excites waves in the small-viscosity limit. Broadside oscillations and the effect of density diffusion are also considered.

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PT Vortex Dynamics and Vortex Flows VI 200H

11:40AM PT.00001 ABSTRACT WITHDRAWN –

11:53AM PT.00002 Trailing vortices from low speed flyers[1] , RYE WALDMAN, JUN KUDO, KENNETH BREUER, Brown University — The structure and strength of the vortex wake behind a airplane or animal flying with a fixed or flapping wing contains valuable information about the aerodynamic load history. However, the amount of vorticity measured in the trailing vortex is not always in agreement with the known lift generated, and the behavior of these vortices at relatively low Reynolds numbers is also not well-understood. We present the results from a series of wind tunnel PIV experiments conducted behind a low-aspect ratio rectangular wing at a chord-Reynolds numbers of 30,000. In addition to wake PIV measurements measured in the cross-stream (Trefftz) plane, we measure the lift and drag directly using a six-axis force-torque transducer. We discuss how vortex size, shape, strength and position vary in time and downstream location, as well as the challenges associated with the use of PIV wake measurements to accurate determine aerodynamic forces.

[1]Supported by AFOSR.

12:06PM PT.00003 Optimal passive pitch reversal via elastic coupling , MICHELE MILANO, ROBERT SPADE, DAVID JURJEVICH, Arizona State University — We consider a prototypical experimental setup, comprising a pitching and heaving rectangular plate. The plate is heaving sinusoidally at constant frequency and variable amplitudes, and a rotational spring generates the pitching motion passively. The rotational spring is simulated by a servo motor driven by a model following controller, and a genetic algorithm optimizes the spring parameters so as to maximize the average lift produced. We present results showing the relationship between optimal parameters for linear and nonlinear springs, and we also investigate the effect of the tip region flow on optimality. PIV measurements for the optimal cases highlight critical features of the flow, showing that the passive pitch reversal is due mainly to leading edge vortices.

12:19PM PT.00004 Evolving Structure of Tip-Vortex Generated by Helicopter Rotor Blade in a Hover , YONG OUN HAN, Yeungnam University, BYUNG HO PARK, YONG JOON SON — In a hover the vortex trail forms a helical trace with respect to the vertical rotor hub axis, and the slip stream becomes contracted rather drastically within one revolution of rotor blade, making the angular momentum of the tip vortex grow quickly. And, as wake age grows, tip vortex experiences diffusion, distortion and stretching during its evolving process. Vortex diffusion proceeds continuously just after matured until disappeared, but distortion and stretching eventuates far downstream with both being correlated. At this stage the helical trace cannot be maintained. Two bladed rotor provides information about vortex distortion as well as diffusion within one revolution of both blades with simplicity. This system makes it possible to observe the change of vortex structure before and after 180 degrees of wake age within one revolution of blade. It is naturally expected that tip vortex affected by the second blade may experience the distortion including the vortex diffusion. This paper aimed primarily to investigate change of vortex structures without and with the second blade effect by the use of experimental devices. It was resulted that tip vortices generated by the first blade satisfy Landgrebe's model of their locations even after they were accelerated by the second blade in downstream. Swirl velocity components follow Vatistas' n=2 model on both regions without loss of vortex circulation.

12:32PM PT.00005 Effect of Liquid Viscosity on the Instability of Polygonal Pattern within a Hollow Vortex Core[1] , AMR MANDOUR, HAMID AIT ABDERRAHMANE, GEORGIOS H. VATISTAS, LYES KADEM, HOI DICK NG, Concordia University — The present study deals with the effect of the viscosity on the speed and limits of endurance of the polygonal pattern, observed within the hollow-core vortex generated by rotating a flat disc near the bottom of a cylindrical tank. Previous investigations have used water, where the Froude number (Fr) appears as the main control parameter. Varying the viscosity of the fluid introduce another independent control parameter, Reynolds number (Re). Using image and signal processing techniques, the influence of these two control parameters on the dynamics of the polygonal pattern is thoroughly investigated. The viscosity of the rotated fluid is varied by mixing glycerol with water and the mixture viscosity is measured using Zahn cup viscometer. The disc rotational speed is measured and controlled accurately by a PID controller loop using LabView environment. A 3-D bifurcation diagram (Fr,Re,N), where the stability region of *N-gon* is delimited, is given. Moreover, the influence of the viscosity on the evolution of the *N-gon* speed is also obtained.

[1]This work is supported by the Natural Sciences and Engineering Research Council of Canada.

12:45PM PT.00006 Origin and stability of radial density stratification in vortical flow , HARISH DIXIT, RAMA GOVINDARAJAN, EMU, JNCASR, Bangalore — A density interface near a vortex winds into a spiral. In the absence of gravity we have shown numerically and through stability analysis that this leads to spiral Kelvin-Helmholtz and centrifugal Rayleigh-Taylor (CRT) instabilities, often reducing the life-time of the vortex dramatically. In this talk we focus on CRT instability, which is driven by centrifugal forces, so flows where the vortex core is heavier than the surroundings are expected to be unstable. Indeed, Sipp et al. (JFM, 2005) and Joly et al. (JFM, 2005), prescribing Gaussian vorticity and density profiles, find no instability in light-cored vortices. Surprisingly however, for some range of parameters, with a Rankine vortex and a step circular density jump we find that making the core lighter can lead to instability. The sharpness of the profiles are thus relevant. The mechanism for this instability, studied by constructing planar analogues in the form of stratified shear flows, bearing similarities with Craik & Adams (JFM, 1979), will be discussed. So will the nonlinear stages of these instabilities from our direct numerical simulations.

12:58PM PT.00007 Search for Euler Singularities with Interacting Vortex Filaments , SAHAND HORMOZ, MICHAEL BRENNER, School of Engineering and Applied Sciences, Harvard University — A possible mechanism for realization of the Euler Singularity is interacting vortex filaments. Numerical simulations, however, have failed to identify initial conditions leading to a singularity. We present recent work on use of similarity solutions to assess the potential for singularity formation for arbitrary initial configurations of interacting vortex filaments.

1:11PM PT.00008 ABSTRACT WITHDRAWN –

1:24PM PT.00009 A Circulation Constraint on Enstrophy Growth in 3D Euler[1] , ROBERT M. KERR, University of Warwick, MIGUEL BUSTAMANTE, University College Dublin — It is shown that by careful consideration of circulation conservation in simulations of anti-parallel Euler vortices how one can discriminate between clean, potentially singular calculations and contaminated, non-singular results. For the latest case with singular behavior it is shown empirically that non-viscous enstrophy growth is consistent with the rigorous upper bound for enstrophy growth for the viscous Navier-Stokes equation, if one replaces viscosity with circulation in the argument. The new scaling laws are consistent with the older numerics presented by Kerr (1993), but many of the conclusions of the old analysis require modification. Besides a new power law for enstrophy growth, if the growth of the maximum of vorticity obeys $\|\omega\|_\infty \sim (T_c - t)^{-\gamma}$, then we find that $\gamma > 1$ and that $\gamma \equiv 1$ as previously reported is unlikely.

[1]Leverhulme Foundation

1:37PM PT.00010 Stability of four point vortices in a periodic strip, the "Domm system"[1] , VASILEIOS VLACHAKIS, ESM Department, Virginia Tech, HASSAN AREF[2], Technical University of Denmark — We approach the modeling of vortex wakes and their stability by considering a system of four vortices, two of circulation $+\Gamma$, two of circulation $-\Gamma$, in a periodic strip, a system first considered by Domm in 1956. The four-degree-of-freedom "Domm system" can be reduced to a system with two degrees of freedom by canonical transformations (Eckhardt & Aref, 1988). The reduced representation allows us to construct perturbations that preserve linear impulse (momentum) and kinetic energy (Hamiltonian) of the system improving upon earlier work by Dolaptschiev and Schmieden from the 1930's. We show that the only translating relative equilibria of the Domm system are the vortex streets that one already finds for two opposite vortices in a periodic strip. We also find by numerical experiments that the vortex street will dissolve into vortex pairs that escape to infinity, a mode of vortex street breakdown observed numerically by Aref & Siggia (1981) and in soap film experiments by Couder & Basdevant (1986). The dissolution process is extremely sensitive to the initial perturbation, suggesting that a form of chaos is involved.

[1]Supported by a Niels Bohr Visiting Professorship from the Danish National Research Foundation.
[2]Permanent address: ESM Department, Virginia Tech

Tuesday, November 24, 2009 11:40AM - 1:24PM –
Session PU Viscous Flows II 200I

11:40AM PU.00001 Colloidal crystal growth by evaporation-induced convective steering , DAMIEN D. BREWER, SATISH KUMAR, MICHAEL TSAPATSIS, University of Minnesota — We simulate evaporation-driven self-assembly of colloidal crystals using an equivalent network model. Relationships between a regular hexagonally close-packed array of hard, monodisperse spheres, the associated pore space, and selectivity mechanisms for face-centered cubic microstructure propagation are described. Accounting for contact line rearrangement and evaporation at a series of exposed menisci, the equivalent network model describes creeping flow of solvent into and through a rigid colloidal crystal. Observations concerning colloidal crystal growth are based on the convective steering hypothesis, which posits that solvent flow into and through the pore space of the crystal may play a major role in colloidal self-assembly. Aspects of the convective steering and deposition of high-Peclet-number rigid spherical particles at a crystal boundary are inferred from spatially resolved solvent flow into the crystal. Gradients in local flow through boundary channels were predicted due to the channels' spatial distribution relative to a pinned free surface contact line. When the free surface contact line is pinned near the leading crystal edge, the network simulations suggest that rows of particles preferentially nucleate furthest from the substrate. These lattice sites propagate the existing microstructure and lead to a declining shelf formation.

11:53AM PU.00002 Motion of a spheroidal capsule in a simple shear flow , JOHANN WALTER, ANNE-VIRGINIE SALSAC, DOMINIQUE BARTHÈS-BIESEL, CNRS - Univ. de Technologie de Compiègne — A capsule is a liquid droplet enclosed in a thin hyperelastic membrane. Microcapsules have various biomedical applications and can be studied as models for circulating cells, especially red blood cells. While many numerical studies have focussed on initially-spherical capsules, the aim of this work is to model the behavior of an ellipsoidal capsule in a simple shear flow. The capsule wall is modeled using finite membrane elements, while the fluids are treated using the boundary integral formulation of the Stokes equations. The viscosity ratio between the inner and outer fluids is kept at 1. When the membrane stiffness is large compared to the flow strength, the capsule undergoes a"tumbling" motion. As the flow rate is increased, a transition occurs toward a "swinging" motion. In this study, we show the influence of the ellipticity of the capsule and of the law modeling the membrane on the behavior of the capsule. Aspect ratios ranging from $1:4$ (oblate) to $4:1$ (prolate) are considered, and two membrane laws (the neo-Hookean law and Skalak's) are compared.

12:06PM PU.00003 Capillary oscillations and periodic defect formation in planar-flow spin casting of molten metal , BRENTON COX, PAUL STEEN, Cornell University — In the planar-flow casting process, surface tension holds liquid metal in a "puddle" where a thin ribbon is solidified as a product. Variations in the final thickness of the ribbon product are typically undesirable in manufacturing and are considered defects. Two similar periodic defects which appear in the ribbon product are studied and will be compared in this talk. The defect with wavelength λ of order 1 mm appears at lower pressures. In higher pressure casts, the defect with λ of order 10 mm appears. While the defect frequencies differ, they are found to scale with the same capillary/inertial time scale. It is observed experimentally that pinning/depinning on the upstream meniscus of the puddle determines which of the two defects will appear. When the upstream meniscus is pinned at the inlet aperture (constrained), the shorter wavelength defect appears. When the meniscus unpinned (free to move), the longer wavelength defect appears in the ribbon product. Instances in which the upstream meniscus may only partially pin due to imposed geometry of the inlet aperture result in coincidence of the two defects.

12:19PM PU.00004 Dynamics of the capillary penetration of a viscous liquid into corrugated walls , FAUSTO SANCHEZ, FIME Universidad Autónoma de Nuevo León, ABRAHAM MEDINA, ESIME Azcapotzalco, IPN, FRANCISCO HIGUERA, ETSI Aeronauticos, UPM — In this work we present a theoretical study of the spontaneous capillary flow, of a viscous liquid, developed into the gap between a couple of parallel corrugated plates (corrugated Hele- Shaw cell). The periodical corrugation of the interior walls of the plates is assumed as a sine-like pattern, transverse to the flow direction. Such a configuration may generate periodical gaps with a structure where zones of maximum and minimum closing occur. This is a simple idealization of typical micro and nano fabricated gaps used to mould polymers by capillarity. By using the lubrication theory we found that a very peculiar temporal flow is developed which could be of interest to improve the knowledge of this type of moulding.

12:32PM PU.00005 Gas flows through shallow microchannel junctions and networks , AMIR GAT, ITZCHAK FRANKEL, DANIEL WEIHS, Technion - Israel Institute of Technology — We study creeping compressible (and incompressible) flows through micro-channel networks whose depth is small in comparison with all other dimensions. Making use of an extended Hele-Shaw asymptotic scheme together with conformal mapping we obtain the relation between the mass-flow-rate and the entrance and exit pressures of a T-junction, thereby quantifying the effects of the junction on the pressure field. The linearity of the problem in terms of an appropriately defined quadratic form of the pressure allows us to apply the results for a single junction towards the optimization of (inter-digitated) networks of parallel-micro-channels which include multiple T-junctions.

12:45PM PU.00006 Numerical Study of the Buoyancy-Driven Flow in a Four-Electrode Rectangular Electrochemical Cell , ZHANYU SUN, VADIM AGAFONOV, CATHERINE RICE, JACOB BINDLER, MET Tech. Inc. — Two-dimensional numerical simulation is done on the buoyancy-driven flow in a four-electrode rectangular electrochemical cell. Two kinds of electrode layouts, the anode-cathode-cathode-anode (ACCA) and the cathode-anode-anode-cathode (CAAC) layouts, are studied. In the ACCA layout, the two anodes are placed close to the channel outlets while the two cathodes are located between the two anodes. The CAAC layout can be converted from the ACCA layout by applying higher electric potential on the two middle electrodes. Density gradient was generated by the electrodic reaction $I_3^- + 2e^- = 3I^-$. When the electrochemical cell is accelerated axially, buoyancy-driven flow occurs. In our model, electro-neutrality is assumed except at the electrodes. The Navier-Stokes equations with the Boussinesq approximation and the Nernst-Planck equations are employed to model the momentum and mass transports, respectively. It is found that under a given axial acceleration, the electrolyte density between the two middle electrodes determines the bulk flow through the electrochemical cell. The cathodic current difference is found to be able to measure the applied acceleration. Other important electro-hydrodynamic characteristics are also discussed.

12:58PM PU.00007 Drag Reduction of a Sphere by Ambient Perturbation and its Relationship with Stokes Layer , MASAYA MUTO, MAKOTO TSUBOKURA, NOBUYUKI OSHIMA, Hokkaido University — An interaction between boundary layer of a sphere and equivalent Stokes layer generated by a perturbation in solid-air two-phase flow has been investigated using numerical simulation. In this simulation, the sphere is fixed in a cylindrical channel. Particle Reynolds number in this study is around 200 that means the wake of the sphere is steady and axisymmetric, and the friction drag and the pressure drag of the sphere are comparable. As a result of simulation, a drag reduction was found in a uniform flow with single-period perturbation oscillating in the same direction as the uniform flow when Stokes layer thickness is smaller than twice of boundary layer thickness in the sphere surface. A reason of the reduction of drag force is a reduction of friction drag caused by that fluctuation energy generated by the perturbation concentrates in the vicinity of particle and the time averaged velocity gradient becomes smaller.

1:11PM PU.00008 The Complex Nature of Turbulence Transition in Boundary Layer Flow , JIM CHEN, CHEN WEIJIA, Nanyang Technological University — Turbulence transition is the process where a laminar field evolves to become turbulent, signaled by the presence of random oscillations. To fully understand turbulence transition demands thoughtful and interconnected consideration of three elements that describe its nature, its physics, mathematics, and numerical simulation. They are interdependent, mutually illuminating elements that form a problem of awe-inspiring complexity. They also examine the problem from a spectrum of perspectives. First, a far-sighted perspective is needed that views the overall problem in its macroscopic, general terms such as its experimental settings, governing equations and boundary conditions. Accompanying it is a precision-oriented, detail-solicitous view of minute, microscopic intricacies such as small-scale turbulence vortices and infinitesimal instability wave interactions that transpire in the flow. The capacity and tolerance to switch between varying degrees of these orthogonally-oriented perspectives is necessary to systematically break down this problem. This study demonstrates a deconstruction of the phenomenon of turbulence transition in order to gain an appreciation of its very complex nature.

Tuesday, November 24, 2009 11:40AM - 1:50PM –

Session PV Micro Propulsion II 205A-D

11:40AM PV.00001 ABSTRACT WITHDRAWN –

11:53AM PV.00002 Simulation of the flow field and tumbling dynamics of multiply flagellated bacterium , RONALD LARSON, NOBUHIKO WATARI, University of Michigan — To study the hydrodynamics of swimming of multi-flagellated bacteria, such as Escherichia coli, we develop a simulation method using a bead-spring model to account for the hydrodynamic and the mechanical interactions between multiple flagella and the cell body, the reversal of the rotation of a flagellum in a tumble and associated polymorphic transformations of the flagellum. This simulation reproduces the experimentally observed behaviors of E. coli, namely, a three-dimensional random-walk trajectory in run-and-tumble motion and steady clockwise swimming near a wall. Here we show using a modeled cell that the polymorphic transformation of flagellum in a tumble facilitates the reorientation of the cell, and that the time-averaged flow field near a cell in a run has double-layered helical streamlines. Moreover, the instantaneous flow field, which is strongly time-dependent, is more than 10-fold larger in magnitude than the time-averaged flow, large enough to affect the migration behavior of surrounding chemoattractants, with the Peclet number for these molecules being larger than one near a swimming cell.

12:06PM PV.00003 A PIV study of the flow field around a rigid rotating helix , SHAN ZHONG, Manchester University, VICTOR PINEDO, ALEXANDER SMITS, Princeton University — Micro-organisms such as bacteria and spermatozoa propel themselves through viscous fluid using flagella which exhibit helical waves. Their typical swimming Reynolds number is 10^{-3} or less. This study presents an attempt of examining the flow field around a rotating flagellum using a scaled-up model. The helical models are made of 2mm diameter metal wires. They have a helical diameter of 17mm and an axial length of 220mm. Three helices with a different pitch angle of 30°, 45° and 60° were tested. The experiment was performed in a rectangular silicone oil bath with a dynamic viscosity of 4.875 Pa.s. The helix was rotated at a frequency of 0.25, 0.5 and 1Hz respectively. The Reynolds number based on the tangential velocity of the rotating helix ranged from 0.02 to 0.09. PIV measurements were undertaken on the central plane parallel to the helical axis. The thrust produced by the rotating helices were also measured using a force sensor.

12:19PM PV.00004 Of cilium and flagellum kinematics[1] , PROMODE R. BANDYOPADHYAY, JOSHUA C. HANSEN, Naval Undersea Warfare Center — The kinematics of propulsion of small animals such as paramecium and spermatozoa is considered. Larger scale models of the cilium and flagellum have been built and a four-motor apparatus has been constructed to reproduce their known periodic motions. The cilium model has transverse deformational ability in one plane only, while the flagellum model has such ability in two planes. When the flagellum model is given a push-pull in one diametral plane, instead of transverse deflection in one plane, it forms a coil. Berg & Anderson's postulation (*Nature* **245** 1973) that a flagellum rotates, is recalled. The kinematics of cilia of paramecium, of the whipping motion of the spermatozoa flagella, and of the flapping motion (rolling and pitching) of the pectoral fins of much larger animals such penguins, have been reproduced in the same basic paramecium apparatus. The results suggest that each of the tiny individual paramecium propulsors have the intrinsic dormant kinematic and structural building blocks to optimize into higher Reynolds number propulsors. A synthetic hypothesis on how small might have become large is animated.

[1]Sponsored by ONR 333 (Optimization Program).

12:32PM PV.00005 External vortex pumping by oscillating plate arrays of mayfly nymphs , ANDREW SENSENIG, Univ. of Akron, KEN KIGER, JEFFREY SHULTZ, Univ. of Maryland — Mayfly nymphs are aquatic insects, many of which can generate ventilation currents by beating two linear arrays of external plate-like gills. The oscillation Reynolds number associated with the gill motion changes with animal size, varying from $Re \sim 2$ to 50 depending on age and species. Thus mayflies provide a novel system model for studying ontogenetic changes in pumping mechanisms associated with transitions from a more viscous- to inertia-dominated flow. Observation of the 3-D kinematics of the gill motion of the species *C. triangulifer* reveal that the mayfly makes a transition in stroke motion when $Re >$5, with a corresponding shift in mean flow from the ventral to the dorsal direction. Time-resolved PIV measurements within the inter-gill space reveal the basic elements of the flow consist of vortex rings generated by the strokes of the individual gills. For the larger Re case, the phasing of the plate motion generates a complex array of small vortices that interact to produce an intermittent dorsally directed jet. For $Re <$5, distinct vortices are still observed, but increased diffusion creates vortices that simultaneously envelope several gills, forcing a new flow pattern to emerge and preventing the effective use of the high Re stroke kinematics. Thus we argue the transition in the kinematics is a reflection of a single mechanism adapted over the traversed Re range, rather than a shift to a completely new mechanism. This work is supported by the NSF under grant CBET-0730907.

12:45PM PV.00006 Regularized Slender Body Theory[1] , MICHAEL NICHOLAS, RICARDO CORTEZ, Tulane University — Various slender body theories allow for the representation of filaments in Stokes' flow by a distribution of fundamental solutions along the filament center line. The idea is revisited here in the more general setting of regularized forces in a small neighborhood along the center line. The regularity in the forces produces a smooth final expression that helps eliminate the computational instabilities of the unregularized formulas. The derivations of the regular slender body theories corresponding with the standard theories of Lighthill and of Keller and Rubinow are outlined. Consistency with these theories is verified in the limit as the smoothing parameter vanishes. Numerical issues of the resulting theories are addressed in the context of test problems.

[1]Work suppoted by NSF VIGRE grant.

12:58PM PV.00007 Simulation of flagellar motions near a rigid surface[1] , RICARDO ORTIZ, RICARDO CORTEZ, Tulane University, MARTIN BEES, University of Glasgow, JOHN KESSLER, LUIS CISNEROS, University of Arizona — Simulations of the hydrodynamic interaction of rotating flagella with a neighboring solid surface are presented using the method of Regularized Stokeslets. We include in the method the use of regularized rotlets and a system of images that exactly cancels the fluid velocity at the wall. The results show features such as an attraction towards the surface and rotations that generate a drag force that allows the flagellum to "roll" along the surface. Other compute flows resemble observed features of the flow when organisms are near the bottom of the plate in an experimental setting.

[1]Sponsored by NSF.

1:11PM PV.00008 The Optimal Elastic Flagellum , SAVERIO SPAGNOLIE, ERIC LAUGA, UC San Diego — We address the question of optimality for slender swimming bodies or flagella in viscous fluid environments. Our novel approach is to define an energy which includes not only the work performed against the surrounding fluid, but also the energy stored elastically in the bending of the body, the energy stored elastically in internal shearing (such as the relative sliding of microtubules internal to a flagellum), and viscous dissipation due to the presence of an internal fluid. The shape of the optimal periodic planar wave is determined numerically and in some cases analytically which maximizes a related efficiency measure. We find that bending or internal dissipation costs regularize the optimal shape, but elastic shearing costs do not. For bodies of finite length, we show that the number of wavelengths expressed by the body is determined by a competition between bending costs and the work done on the fluid associated with body rotations. The hydrodynamic efficiency is shown to be less sensitive to the morphology than the bending costs, which may help us to better understand the locomotory forms observed in nature.

1:24PM PV.00009 A mechanism of low-Reynolds-number propulsion enhancement , ALEXANDER LESHANSKY, Department of Chemical Engineering, Technion-IIT, Haifa 32000 — Is has been known for some time that some microorganisms can swim faster in high-viscosity gel-forming polymer solutions. The qualitative explanation of this phenomena first offered by H. Berg and L. Turner (Nature **278**, 349, 1979) suggested that propulsion enhancement is a result of flagellum pushing on quasi-rigid loose polymer network. Following this hypothesis we consider inertia-less propulsion in a model heterogeneous environment composed of sparse array of stationary obstacles embedded into a viscous Newtonian solvent. It is demonstrated that for some propulsion techniques, including transverse surface waves and rotating helical filament, the propulsion speed (for the prescribed swimming gait) is enhanced when compared to the locomotion through viscous solvent. It is shown that locomotion is not only advantageous speed-wise, but is also more hydrodynamically efficient. The results of the rigorous numerical simulations of the rotating "shish- kebab" filament propelled through a random sparse matrix of stationary spherical obstructions are in a very close agreement with the predictions of proposed modified resistive force theory based on effective media approximation.

1:37PM PV.00010 Unsteady low-Re swimming , ON SHUN PAK, ERIC LAUGA, University of California San Diego — In this talk, we focus on unsteady effects relevant to the fluid-based locomotion of micro-organisms. First, we consider transient effects in locomotion arising from the inertia of both the swimmer and the surrounding fluid. We discuss and derive the relevant time scales governing transient effects in low Reynolds number swimming, and illustrate them using the prototypical problem of a 2D swimmer starting from rest. Second, we address geometrical unsteadiness resulting from the finite-size of the swimmer. We solve numerically for the swimming kinematics of active (internally-forced) filaments, as models for eukaryotic flagella, and discuss the resulting unsteadiness of the cell body.

Tuesday, November 24, 2009 11:40AM - 1:50PM –
Session PW Biofluids XI: Cellular III - Viscoelastic Biofluids 208A-D

11:40AM PW.00001 The dynamics of immersed boundaries in viscoelastic fluids , JOHN CHRISPELL, RICARDO CORTEZ, DAMIR KHISMATULLIN, LISA FAUCI, Tulane University — Many biological fluids are viscoelastic and require a nonlinear constitutive equation to describe the evolution of the extra-stress tensor. We use an immersed boundary framework to model processes that involve the movement of immersed elastic boundaries interacting with a surrounding viscoelastic fluid. We present recent results on applications including dynamics of a closed membrane moving under surface tension, and phase-locking of swimming sheets.

11:53AM PW.00002 Buckled in translation , ANKE LINDNER, ESPCI, ELIE WANDERSMANN, Leiden University, NAWAL QUENNOUZ, OLIVIA DU ROURE, ESPCI, YUAN-NAN YOUNG, NJIT, MICHAEL SHELLEY, Courant Institute, NYU — An elastic filament can undergo a buckling instability when interacting with a viscous flow. The subsequent deformation of the filament changes its transport properties in the flow. In particular, numerical simulations (Young *et al.* Phys. Rev. Lett.,99,058303, 2007) have shown that due to its deformability an elastic filament can move like a random walker in a cellular flow, formed by closed stream lines. We have built an experimental set up using a centimeter scale filament made from a silicon elastomer in a network of counter rotating vortices allowing for a direct study of the coupling of deformation and transport. We quantify the buckling threshold in the complex flow geometry and show that it is in good agreement with analytical and numerical predictions. We directly link the observed buckling to modifications in the transport of the filament and study the probability of the filament to escape from a given vortex as a function of its deformability. We furthermore compare the transport in the vortex array of a rigid and a flexible filament to a small bead and show that not only deformability but also the size of the filament plays a crucial role for its transport behaviour.

12:06PM PW.00003 The Stochastic and Driven Dynamics of Microscopic Elastic Objects Coupled by a Viscous Fluid , MATTHEW CLARK, MARK PAUL, Virginia Tech — We investigate analytically and numerically the coupled motion of microscopic objects in a viscous fluid. Fluid-coupled structures are encountered across a broad range of fields, including spheres and cantilevers in microscopic instruments and fluid motion sensing in biological systems. Many small scale systems undergo high frequency oscillations with small magnitude resulting in a flow field with significant local inertia contributions that must be described using the unsteady Stokes equation. We study the fluid coupled motion of two infinite cylinders that are each attached to a spring. This geometry is chosen due to its wide use in modeling cantilevers and beams in fluid. We show that the stochastic and driven correlated motion of the two cylinders can be found from a single deterministic calculation – the response of the cylinders to an impulse in force. The stochastic dynamics are found using the fluctuation-dissipation theorem and the driven dynamics are found using transfer function theory. We derive analytical expressions for the cylinder dynamics that neglects effects of back-action. We compare our analytical expressions with finite element numerical simulations and find our analysis is valid over a range of larger separations. For small separations, with overlapping Stokes layers, we find interesting variations in both the amplitude and phase of the cylinders.

12:19PM PW.00004 Affine transformations capture beak shape variation in Darwin's Finches , MICHAEL BRENNER, OTGER CAMPAS, School of Engineering and Applied Sciences, Harvard University, RICCARDO MALLARINO, ARHAT ABZHANOV, Department of Organismic and Evolutionary Biology, Harvard University — Evolution by natural selection has resulted in extraordinary morphological complexity of living organisms, whose description has thus far defied any precise mathematical characterization linked to the underlying developmental genetics. Here we demonstrate that the morphological diversity of the beaks of Darwin's finches, the classical example of adaptive morphological radiation, is quantitatively accounted for through the mathematical group of affine transformations. Specifically, we show that all beak shapes of Ground Finches (genus *Geospiza*) are related by scaling transformations (a subgroup of the affine group), and the same scheme occurs for all the beak shapes of Tree and Warbler finches. This analysis shows that the beak shapes within each of these groups differ only by their scales, such as length and depth, each of which is knownto be under genetic control.The complete morphological variability within the beaks of Darwin's finches can be explained by extending the scaling transformations to the entire affine group, by including shear transformations. Altogether our results suggest that the mathematical theory of groups can help decode morphological variability, and points to a potentially hierarchical structure of morphological diversity and the underlying developmental processes.

12:32PM PW.00005 An integrated muscle mechanic-fluid dynamic model of lamprey swimming , CHIA-YU HSU, Tulane University, ERIC TYTELL, University of Maryland, LISA FAUCI, Tulane University — In an effort towards a detailed understanding of the generation and control of vertebrate locomotion, including the role of the CPG and its interactions with reflexive feedback, muscle mechanics, and external fluid dynamics, we study a simple vertebrate, the lamprey. Lamprey body undulations are a result of a wave of neural activation that passes from head to tail, causing a wave of muscle activation. These active forces are mediated by passive structural forces. We present recent results from a model that fully couples a viscous, incompressible fluid with nonlinear muscle mechanics. We measure the dependence of the phase lag between activation wave and mechanical wave as a function of model parameters, such as body stiffness and muscle strength. Simulation results are compared to experiments utilizing both real and synthetic lamprey.

12:45PM PW.00006 Electro-hydrodynamic effects on lipid membranes in giant vesicles , MARGARITA STAYKOVA, TETSUYA YAMAMOTO, REINHARD LIPOWSKY, RUMIANA DIMOVA, Max Planck Institute of Colloids and Interfaces, Department of Theory and Biosystems, Potsdam, Germany — Electric fields are widely applied for cell manipulation in numerous micron-scale systems. Here, we show for the first time that alternating electric fields may cause pronounced flows in the membrane of giant lipid vesicles as well as in the surrounding fluid media.The lipid vesicles are not only biomimetic model for the cell membrane but also have many potential biotechnological applications, e.g. as drug-delivery systems and micro-reactors. The reported effects should be considered in electric micro-manipulation procedures on cells and vesicles. They might be useful for applications in microfluidic technologies, for lipid mixing, trapping and displacement, as will be demonstrated. We also believe that our method for visualization of the lipid flows by fluorescently labeled intra-membrane domains will be helpful for studies on membrane behavior in vesicles subjected to shear or mechanical stresses.

12:58PM PW.00007 Elastohydrodynamics of wet bristles, carpets and brushes , LAKSHMINARAYANAN MAHADEVAN[1], SEAS, Harvard University, ARVIND GOPINATH, Mechanical Engineering, MIT — We present an effective field theory for the elastohydrodynamics of ordered brushes and disordered carpets. These soft beds are comprised of elastic filamentous units, interspersed in a fluid and grafted on a substrate. Our formulation leads naturally to a set of constitutive equations coupling bed deformation to fluid flow, accounts for anisotropic properties of the medium, and generalizes poroelasticity to these systems. These effective medium equations are then used to study two canonical problems - the normal settling of a rigid sphere onto a carpet, and the tangential shearing motion of a rigid sphere over the carpet, both problems of much relevance in mechanosensation in biology.

[1]Professor, SEAS. Affiliate Professor, Systems Biology (Harvard Medical School)

1:11PM PW.00008 Dynamics of polarly driven filaments[1] , MICHAEL SHELLEY, New York University, YUAN-NAN YOUNG, New Jersey Institute of Technology — In this study we investigate the dynamics and transport of bio-polymers such as microtubules or actin filaments when driven by motor proteins. A slender-body hydrodynamic formulation is augmented by a model for the forcing from the motor proteins. Our results show that the spatial gradients in the polar forcing, or the detachment of the filament from the motor proteins, can lead to nontrivial shape dynamics (undulatory traveling waves), and random walk trajectories. The interaction between filament and the simple device geometries is investigated, as is the effect of thermal fluctuations.

[1]Support from NSF is acknowledged

1:24PM PW.00009 A fiber-reinforced-fluid model of anisotropic plant root cell growth[1] , OLIVER E. JENSEN, School of Mathematical Sciences, University of Nottingham, UK, ROSEMARY J. DYSON, Centre for Plant Integrative Biology, University of Nottingham, UK — We present a theoretical model of a single cell in the expansion zone of the primary root of the plant *Arabidopsis thaliana*. The cell undergoes rapid elongation with approximately constant radius. Growth is driven by high internal turgor pressure causing viscous stretching of the cell wall, with embedded cellulose microfibrils providing the wall with strongly anisotropic properties. We represent the cell as a thin cylindrical fiber-reinforced viscous sheet between rigid end plates. Asymptotic reduction of the governing equations, under simple sets of assumptions about fiber and wall properties, yields variants of the traditional Lockhart equation that relates the axial cell growth rate to the internal pressure. The model provides insights into the geometric and biomechanical parameters underlying bulk quantities such as wall extensibility and shows how either dynamical changes in wall material properties or passive fibre reorientation may suppress cell elongation.

[1]The Centre for Plant Integrative Biology is supported by BBSRC and EPSRC.

1:37PM PW.00010 Effects of flow on insulin fibril formation at an air/water interface , DAVID POSADA, CARYN HELDT, MIRCO SORCI, GEORGES BELFORT, AMIR HIRSA, Rensselaer Polytechnic Institute — The amyloid fibril formation process, which is implicated in several diseases such as Alzheimer's and Huntington's, is characterized by the conversion of monomers to oligomers and then to fibrils. Besides well-studied factors such as pH, temperature and concentration, the kinetics of this process are significantly influenced by the presence of solid or fluid interfaces and by flow. By studying the nucleation and growth of a model system (insulin fibrils) in a well-defined flow field with an air/water interface, we can identify the flow conditions that impact protein aggregation kinetics both in the bulk solution and at the air/water interface. The present flow system (deep-channel surface viscometer) consists of an annular region bounded by stationary inner and outer cylinders, an air/water interface, and a floor driven at constant rotation. We show the effects of Reynolds number on the kinetics of the fibrillation process both in the bulk solution and at the air/water interface, as well as on the structure of the resultant amyloid aggregates.