64th Annual Meeting of the APS Division of Fluid Dynamics

Baltimore, Maryland

http://www.aps.org/meetings/meeting.cfm?name=DFD11

Sunday, November 20, 2011 8:00AM - 10:10AM –
Session A1 Geophysical Flows: Atmospheric I 301

8:00AM A1.00001 Vortex Roll Breakup in Three-Dimensional Turbulent Boundary Layers[1]

CURTIS HAMMAN, PARVIZ MOIN, Center for Turbulence Research, Stanford University — Large helical vortex rolls with axes in the general direction of the mean wind commonly appear in the unstably stratified atmospheric boundary layer. When a rapid shift in the mean wind direction occurs, the vertical transport of momentum and heat flux is sharply reduced compared to the equilibrium value. At long times, this non-equilibrium turbulent flow may develop back into a stable pattern of organized vortex rolls, now aligned with the new wind direction. This transition process is studied via direct numerical simulation of plane channel flow heated from below with impulsively started transverse pressure gradient ($Ri = -Ra/PrRe^2 = -0.25$, $Ra = 10^7$, and $Pr = 0.71$). The timescale for heat flux recovery is approximately the same for turning angles larger than 30 degrees. For higher turning angles, however, the Nusselt number will temporarily drop below one due to a significant reduction in vertical transport. Horizontal velocity and temperature spectra suggest that scale separation between large-scale, organized convective motions and turbulent eddies can prevent heat transfer reduction in transversely accelerated three-dimensional turbulent boundary layers.

[1]Supported by the DOE CSGF, grant no. DE-FG02-97ER25308

8:13AM A1.00002 A vortex pair near a density gradient interface

, SURUPA SHAW, NICK JENKINS, JOHN MCHUGH, University of New Hampshire — The dynamics of a vortex pair in a stratified atmosphere near a density gradient interface is considered here using direct numerical simulations. A density-gradient interface has continuous density but discontinuous gradient of density, and is a common model of the tropopause. The vortex pair is released below the interface and allowed to propagate vertically toward the interface. The anelastic approximation of the Navier-Stokes equations are treated using a spectral method, and the initial vortex has a Gaussian distribution of vorticity. The results show that strong vortices propagate through the interface without much change in dynamics. Weaker vortices will dissipate energy when they reach the interface and although a remnant of the vortex pair transits the interface, it does not achieve the same altitude that it would have without the interface. Overall, the interface is not a barrier to vortex pairs, but would be expected to change the distribution of energy in more complicated flows.

8:26AM A1.00003 3D Vortices in Stratified, Rotating Flows - Secondary Circulations and Changes in Aspect Radio Due to Dissipation

, PHILIP MARCUS, PEDRAM HASSANZADEH, UC Berkeley — The aspect ratio of a 3D vortex in a rotating, stratified flow is defined as the ratio of its vertical half-thickness H to its horizontal scale L. We recently showed that due to hydrostatic and geo/cyclostrophic balance, an anticyclone has an equilibrium scaling law of $H/L = Ro(1 - Ro)f/(\bar{N} - N_{in})$, where Ro is the Rossby number of the vortex, f is the Coriolis parameter, and $\bar{N}$ and N_{in} are the Brunt-Väisälä frequencies of the local ambient fluid and of the vortex interior, respectively. Introduction of a viscous or thermal dissipation (the latter being much more rapid and therefore much more relevant in atmospheric, astrophysical, and planetary vortices) forces a vortex that was initially in equilibrium to decay through a series of quasi-stationary states. Both viscous and thermal dissipation rapidly induce secondary circulations within the vortex, but the circulations created by the two types of dissipation differ qualitatively from each other. Moreover, thermal dissipation rapidly changes the values of Ro and N_{in}, so although the equilibrium scaling law above is still satisfied, the aspect ratio of the vortex changes rapidly. We show how the resulting aspect ratio of the vortex, and the magnitude and geometry of the secondary circulation are both strong functions of the vertical dependence of $\bar{N}$.

8:39AM A1.00004 Large-eddy simulation of the nighttime boundary layer over the US Great Plains[1]

, FOTINI CHOW, BOWEN ZHOU, University of California, Berkeley — The nighttime stable boundary layer (SBL) is associated with various atmospheric processes including low-level jets, internal gravity waves, Kelvin-Helmholtz shear instabilities, and turbulence events. These phenomena due to the strong stable stratification are quite difficult to represent in numerical models and often require very high grid resolution. In this study, nested large eddy-simulations (LES) are performed over the site where the Cooperative Atmospheric-Surface Exchange Study (CASES-99) field experiment took place, near Leon, Kansas. The night of Oct 5-6 (IOP2) is chosen to represent a typical intermittently turbulent night over the Great Plains. Two turbulent bursting events in the SBL are identified. Simulations are initialized with North American Regional Reanalysis (NARR) on 32 km grids, and one-way nested to a very fine grid with 16 m horizontal spacing. Results using the conventional TKE-1.5 and dynamic reconstruction turbulence models are compared with observations. While both turbulence closures predict the first event with great precision at 16 m scale, only the dynamic reconstruction model (DRM) is able to sustain intermittent turbulence and predict the second bursting event.

[1]supported by NSF grant ATM-0645784

8:52AM A1.00005 Large-eddy simulation of stable atmospheric boundary layers to develop better turbulence closures for climate and weather models

, ELIE BOU-ZEID, JING HUANG, Princeton University, JEAN-CHRISTOPHE GOLAZ, Geophysical Fluid Dynamics Laboratory — A disconnect remains between our improved physical understanding of boundary layers stabilized by buoyancy and how we parameterize them in coarse atmospheric models. Most operational climate models require excessive turbulence mixing in such conditions to prevent decoupling of the atmospheric component from the land component, but the performance of such a model is unlikely to be satisfactory under weakly and moderately stable conditions. Using Large-eddy simulation, we revisit some of the basic challenges in parameterizing stable atmospheric boundary layers: eddy-viscosity closure is found to be more reliable due to an improved alignment of vertical Reynolds stresses and mean strains under stable conditions, but the dependence of the magnitude of the eddy viscosity on stability is not well represented by several models tested here. Thus, we propose a new closure that reproduces the different stability regimes better. Subsequently, tests of this model in the GFDL's single-column model (SCM) are found to yield good agreement with LES results in idealized steady-stability cases, as well as in cases with gradual and sharp changes of stability with time.

9:05AM A1.00006 Evening Transition of the Atmospheric Boundary Layer (ABL)

, H.J.S. FERNANDO, University of Notre Dame — Laboratory experiments and theoretical studies were conducted to understand the decay of convective turbulence, with applications to evening transition of ABL over flat terrain. Either a linearly stratified or a homogeneous fluid was used. Initial convection was established with a steady heat flux at the bottom, which was then subjected to gradual cooling. The convection starts to decay with decreasing heat flux, and in stratified runs convective turbulence strongly interacts with the stratification aloft, thus changing the decay behavior substantially depending on the bulk Richardson number. Surprisingly, the turbulence modification during decay is realized by the adjustments occurring in the u-component rather than the w-component as expected. It is shown that stratification near the bottom surface and strong buoyancy effects in the proximity of entraining inversion suppress vertical motions therein to the extent that horizontal layering and a "jet-like" flow structure form. Enhanced dissipation in these layers leads to rapid decay of turbulent kinetic energy, after a time of about $t = (3\text{-}4)t*$, where $t*$ is the convective time scale at the onset of cooling. Oscillations of velocity and temperature fluctuations, development of layering, influence of overlying stratification and lack of agreement with previous numerical modeling were some of the salient features of experimental results.

9:18AM A1.00007 Experimental Characterization of Radiation Forcing due to Atmospheric Aerosols

, K.R. SREENIVAS, D.K. SINGH, V.K. PONNULAKSHMI, G. SUBRAMANIAN, Engineering Mechanics Unit, JNCASR, Bangalore — Micro-meteorological processes in the nocturnal atmospheric boundary layer (NBL) including the formation of radiation-fog and the development of inversion layers are controlled by heat transfer and the vertical temperature distribution close to the ground. In a recent study, it has been shown that the temperature profile close to the ground in stably-stratified, NBL is controlled by the radiative forcing due to suspended aerosols. Estimating aerosol forcing is also important in geo-engineering applications to evaluate the use of aerosols to mitigate greenhouse effects. Modeling capability in the above scenarios is limited by our knowledge of this forcing. Here, the design of an experimental setup is presented which can be used for evaluating the IR-radiation forcing on aerosols under either Rayleigh-Benard condition or under conditions corresponding to the NBL. We present results indicating the effect of surface emissivities of the top and bottom boundaries and the aerosol concentration on the temperature profiles. In order to understand the observed enhancement of the convection-threshold, we have determined the conduction-radiation time constant of an aerosol laden air layer. Our results help to explain observed temperature profiles in the NBL, the apparent stability of such profiles and indicate the need to account for the effect of aerosols in climatic/weather models.

9:31AM A1.00008 Fluctuations in Atmospheric Boundary Layer Plumes

, JAY BORIS, DAVID FYFE, Naval Research Laboratory, MIYOUNG OBENSCHAIN, Berkeley Research Associates, KEITH OBENSCHAIN, GOPAL PATNAIK, Naval Research Laboratory — Pollution and short-term health considerations require the accurate prediction of airborne contaminant transport in cities. Even when a stationary source emits tracer gases continuously, the resulting plume fluctuates vigorously in the turbulence that results from air passing over any typical landscape. Computing this flow properly requires large-eddy simulations that resolve the vortices shed from buildings, trees, and terrain because these coherent effects govern the "turbulent" dispersion of pollutants, tracer gases, and potentially toxic agents. This paper uses long-time, high-resolution detailed studies of one urban configuration, computed with 5-meter spatial resolution and sub-second temporal resolution, to quantify the deviations of passive tracer plumes from steady state. Even when concentration values at a point are averaged over long times, as an accumulating sensor might do, the range of probable values spans orders of magnitude. At a 5-km scale, averaging tracer concentrations for as long as an hour still leaves likely sampling fluctuations of plus or minus a factor of ten from the long-time average.

9:44AM A1.00009 RANS modeling of scalar dispersion from localized sources within a simplified urban-area model

, RICCARDO ROSSI, STEFANO CAPRA, Università di Bologna, GIANLUCA IACCARINO, Stanford University — The dispersion of a passive scalar downstream a localized source within a simplified urban-like geometry is examined by means of RANS scalar flux models. The computations are conducted under conditions of neutral stability and for three different incoming wind directions (0°, 45°, 90°) at a roughness Reynolds number of Ret = 391. A Reynolds stress transport model is used to close the flow governing equations whereas both the standard eddy-diffusivity closure and algebraic flux models are employed to close the transport equation for the passive scalar. The comparison with a DNS database shows improved reliability from algebraic scalar flux models towards predicting both the mean concentration and the plume structure. Since algebraic flux models do not increase substantially the computational effort, the results indicate that the use of tensorial-diffusivity can be promising tool for dispersion simulations for the urban environment.

9:57AM A1.00010 Dispersion of Particles Emitted from Area Sources into the Unstable Atmospheric Boundary Layer

, YING PAN, MARCELO CHAMECKI, SCOTT ISARD, Pennsylvania State University — Dispersion of heavy particles emitted from area sources into the atmospheric boundary layer is of broad applications in diverse fields. Chamecki and Meneveau (JFM, 2011) used large-eddy-simulation (LES) to study properties of the particle plumes in the neutrally stratified atmosphere, providing theoretical expressions to predict mean particle concentration profiles, plume height, and horizontal transport above the source and ground deposition flux downwind. Here effects of unstable temperature stratification are considered. LES results suggest that particle plume heights, horizontal mass fluxes, and particle loadings downwind from the source increase as the atmospheric instability increases. The Obukhov length and the free-convection velocity are introduced to characterize the deposition fluxes in addition to friction and settling velocities used in the neutral case. The new theoretical predictions that consider the local atmospheric instability in the expression of the vertical velocity variance agree well with LES results when the particle plume is within the surface layer, where the assumption of self-preservation of concentration profiles holds.

Sunday, November 20, 2011 8:00AM - 10:10AM –

Session A2 Granular Flows I 302

8:00AM A2.00001 Predictive Capabilities of a Relaxation Model for Parcel-Based Granular Flow Simulations[1]

, STEFAN RADL, SANKARAN SUNDARESAN, Department of Chemical and Biological Engineering, Princeton University — Parcel-based methods have a great potential to reduce the computational cost of particle simulations for dense flows. Here we investigate a relaxation model, similar to that of Bhatnagar-Gross-Krook (BGK), when applied to such a parcel-based simulation method. Specifically, we have chosen the simulation methodology initially proposed by Patankar and Joseph [1], and combined it with the relaxation model published by O'Rourke and Snider [2]. We show that a relaxation model is key to correctly predicting macroscopic flow features, e.g., the scattering pattern of a granular jet impinging on a flat surface, studied experimentally by Cheng et al. [3]. Simple shear flow simulations reveal that calculation of the locally-averaged velocity is a critical ingredient to correctly predict streaming and collisional stresses.

[1] N.A. Patankar, and D.D. Joseph, Int. J. Multiphase Flow 27, 1659 (2001).
[2] P.J. O'Rourke, and D.M. Snider, Chem. Eng. Sci. 65, 6014 (2010).
[3] X. Cheng et al., Phys. Rev. Lett. 99, 188001 (2007).

[1]SR acknowledges the support of the Austrian Science Foundation through the Erwin-Schroedinger fellowship J-3072.

8:13AM A2.00002 Three dimensional imaging of soft sphere packings under shear[1]

, ROBERT BEHRINGER, JOSHUA DIJKSMAN, ERIC SIA, Duke University — The (microscopic) flow of three dimensional disordered athermal granular packings remains poorly understood. However, experimentally studying flow and deformations in a three dimensional packing of grains is challenging due to the opacity of such packings. Our goal is to study triaxial shear of granular materials, using refractive index matched scanning. We will present results on a study of the deformation of a three dimensional soft sphere packing under quasi static compression. The spheres are made from hydrogel and virtually frictionless, similar to the study by by Mukhopadhyay et. al. (2011). We track particles and image contact deformations, and look at the effect of cyclic shear flow.

[1]NSF-DMR0906908, ARO-W911NF-11-1-0110

8:26AM A2.00003 Emergence of Order in Sheared Systems of Elliptical Particles[1] , SOMAYEH FARHADI, Duke University, Department of Physics, ROBERT P. BEHRINGER, Duke University — We have studied the emergence of order in shear-jammed systems composed of quasi-2D elliptical particles, using both biaxial and Couette shear. The particles are photoelastic ellipses with aspect ratio 2. Using photoelastic particles enables us to determine grain-scale forces, and hence the microscopic force structure of the system. We obtain forces by solving the inverse photoelastic problem for ellipses subject to boundary point forces, similarly to the approach that we have developed for circular particles. We will present data for local packing fraction, orientation and force. Using configurational and force data shows that local smectic order plays a crucial role in jamming of elliptical particles.

[1]Acknowledgement: NSF Grant DMR0906908.

8:39AM A2.00004 Cyclic Simple Shear in a Two-Dimensional Granular System[1] , JIE REN, JOSHUA DIJKSMAN, ROBERT BEHRINGER, Duke University — We study the irreversibility transition observed by Pine et al (2005) in a 2D dry granular system, for which our novel experimental apparatus can create quasi-static, quasi-uniform simple shear. We use bi-disperse, photo-elastic disks and expose them to a large number of small amplitude shear cycles. We track the particles dynamics over time and evaluate their translational and rotational motion, as well as the inter-particle forces. We have found that both translational and rotational diffusivity increase rapidly when the system's particle density exceeds a certain threshold. This is an indication of the existence of an irreversibility transition. Interestingly, we have found this density threshold to be dependent on the amplitude of cyclic shear, but always smaller than the density for the isotropic jamming transition. To further understand this irreversibility transition, we analyze the system's structural and force evolution.

[1]NSF grants: DMR0906908, DMS0835571, NASA grant NNX10AU01G, and ARO grant W911NF-11-1-0110

8:52AM A2.00005 High-velocity drag friction in dense granular media[1] , YUKA TAKEHARA, KO OKUMURA, Department of Physics, Graduate school of Ochanomizu University — We study drag force acting on an obstacle in granular media, focusing on a high-velocity region where only a few direct studies are available. The granular media are two-dimensional and consist of small-sphere particles. A larger-disk obstacle moves in the medium at different constant speeds. The drag force is found to be proportional to the square of the velocity. This clear experimental relation is convincingly explained by a simple and original theory. A crucial assumption of this theory is supported by image analysis for the size of the cluster of grains around the disk. As a result, we conclude that the friction we observed here is physically different from a hydrodynamic inertial friction, which has been discussed in dilute granular flows and in impact experiments. Furthermore we demonstrate how the drag force depends on packing fraction to discuss the possibility of a dynamical jamming transition in our drag experiment.

[1] Y.Takehara, S. Fujimoto and K. Okumura, High-velocity drag friction in dense granular media, *EPL*, **92** (2010) 44003

[1]We thank S. Fujimoto for her contribution at the early stage of this work.

9:05AM A2.00006 Strange eigenmodes of chaotic granular flow in a tumbler[1] , IVAN C. CHRISTOV, JULIO M. OTTINO, RICHARD M. LUEPTOW, Robert R. McCormick School of Engineering and Applied Science, Northwestern University — Through a combined computational–experimental study of monodisperse granular flow in a slowly-rotating quasi-two-dimensional container we show the presence of naturally-persistent mixing patterns, i.e., "strange" eigenmodes of the advection- diffusion operator governing the mixing process in the Eulerian frame. A comparative analysis of the structure of eigenmodes and the corresponding Poincaré section and finite-time Lyapunov exponent field of the flow highlights the relationship between the Eulerian and Lagrangian descriptions of mixing. In addition, we show how the mapping method for scalar transport can be modified to include diffusive effects, which are more significant in a granular flow (in laboratory size equipment) than in a similar fluid flow. This allow us to examine (for the first time in a granular flow) the change in shape, lifespan, and eventual decay of eigenmodes due to diffusive effects at larger numbers of revolutions. Finally, it is shown that segregation patterns in bidisperse mixtures correspond to permanently-excited eigenmodes.

[1]This work was supported, in part, by NSF Grant CMMI-1000469.

9:18AM A2.00007 Pattern formation and pinning-depinning transition in a subharmonic granular wave pattern[1] , CLAUDIO FALCÓN, JEREMíAS GARAY, MARCEL CLERC, IGNACIO ORTEGA, Departamento de Física, Facultad de Ciencias Físicas y Matemáticas, Universidad de Chile — We present an experimental and theoretical study of the pattern formation process of standing subharmonic waves in a fluidized quasi-one-dimensional shallow granular bed. The fluidization process is driven by means of a time-periodic air flow, analogous to a tapping type of forcing. Measurements of the amplitude of the critical mode close to the transition are in quite good agreement with those inferred from a universal stochastic amplitude equation. This allows us to determine both the bifurcation point of the deterministic system and the corresponding noise intensity. Then, we charaterize the stationary to drifting transition of this subharmonic wave pattern in the presence of inhomogeneities and drift forces. The transition is mediated by the competition of the inherent periodicity of the subharmonic pattern, the asymmetry of the system and the finite size of the cell. We measure the mean phase velocity of the subharmonic pattern, which is in good agreement with those inferred from an amplitude equation taking into account asymmetry and finite size effects of the system.

[1]Funded by FONDECYT grants 1090045 and 11090049, and CONICYT grant ACT127.

9:31AM A2.00008 The Dynamics of Granular Materials , MIROSLAV KRAMAR, Rutgers, KONSTANTIN MISCHAIKOW, LOU KONDIC, ARNAUD GOULLET, NJIT — We will present a novel approach to study force chain structures of particulate systems. We concentrate on systems undergoing compression. Our method distinguishes different types of friction as well as different rate of poly dispersity. The topological measures can be also used to understand the dynamic features of the system and correlate these measures to phenomena such as jamming.

9:44AM A2.00009 Theory for shear-induced segregation of dense granular mixtures , YI FAN, KIMBERLY HILL, Department of Civil Engineering, University of Minnesota — It is well-known that a mixture of different sized particles will segregate in a gravitational field. However, it has only recently been shown that a gradient of shear rate alone can drive segregation in dense sheared systems. In contrast with sparse energetic granular materials, in dense sheared systems, large particles segregate to the regions with higher shear rates. We develop a model for shear-induced segregation in dense mixtures of different sized particles. The model is comprised of two primary parts. The first involves the tendency of a gradient in kinetic stress – stress associated with velocity fluctuation correlations – to drive all particles toward regions of low shear rate. The second is essentially a kinetic sieving effect in which small particles are more likely to find voids into which they can travel than large particles. The two features together segregate small particles to regions of low shear rate and squeeze large particles in the opposite direction. We validate this model via 3D Discrete Element Method (DEM) simulations in a vertical chute.

9:57AM A2.00010 Complex Dynamics of a Floating Sheared Granular Layer[1] , QUENTIN SHERMAN, ZACHARY NEEDELL, Haverford College, DOUGLAS DURIAN, University of Pennsylvania, JERRY GOLLUB, Haverford College — We investigate the complex shear flow dynamics of a two-dimensional layer of cohesive particles floating on a magnetically driven fluid. Capillary forces provide cohesion, which can be adjusted using a surfactant. The layer shows a yield force that rises with packing fraction and cohesion. Using a Voronoi tessellation of the particle positions, we find that the distribution of the local free areas follows a Gamma distribution with an additional tail resulting from voids. The deformation gradient of the related Delaunay triangulation of the particle positions provides a useful way to characterize the dynamics. We focus especially on the rotation and stretching rates of these triangles, which are found to be only weakly correlated with the local free area. It appears that local structural quantities cannot reliably predict the evolving regions of large deformation. We note asymmetries of these fluctuations that correspond to elongated regions of high deformation (e.g.slip lines). Finally, we consider the mean square non-affine particle displacements, which appear to grow diffusively in time.

[1]Supported by NSF Grant DMR-1104705

Sunday, November 20, 2011 8:00AM - 10:10AM –
Session A3 Multiphase Flow I: Bubbly Flows 303

8:00AM A3.00001 Bubble Interactions in Multiphase Turbulent Channel Flows[1] , JIACAI LU, Worcester Polytechnic Institute, GRETAR TRYGGVASON, University of Notre Dame — Direct numerical simulations of deformable bubbles in weakly turbulent upflows in vertical channels have shown that the flow has a remarkably simple structure. For upflow, lift forces drive nearly spherical bubbles to the walls, forming a bubble-rich wall-layer. For downflow the bubbles are, on the other hand, driven away from the walls, creating a bubble free wall-layer. In both cases the addition or removal of bubbles to the center continues until the two-phase mixture there is in hydrostatic equilibrium. The lift on deformable bubbles is, however, generally nearly zero and the bubbles do not migrate laterally, on the average. Here we examine the effect of the size of the computational domain, and the number of bubbles, as well as the effect of including bubbles of different sizes. The results show that the distribution of bubbles sliding along the walls in upflow is generally very uneven, with parts of the wall crowded with bubbles and other parts bubble free. The presence of larger bubbles also tends to cause large fluctuations in the flow, including disrupting the wall-layers.

[1]Research supported by DOE (CASL).

8:13AM A3.00002 Effect of Bubble Deformability in Multiphase Turbulent Channel Flows[1] , SADEGH DABIRI, University of Notre Dame, JIACAI LU, Worcester Polytechnic Institute, GRETAR TRYGGVASON, University of Notre Dame — Recent results of direct numerical simulations (DNS) of deformable bubbles in turbulent upflows in vertical channels are discussed. Earlier results have shown that the lift forces leading to a lateral migration of nearly spherical bubbles controls the flow. In weakly turbulent flows, nearly spherical bubbles form a dense bubbly wall-layer but more deformable bubbles stay away from the walls, resulting in very different flow structures and flow rates. The different flow regimes have a very simple structure, but it is necessary to examine the collective motion of many bubbles rather than individual ones to predict the void fraction profile. Here the transition from one flow regime to the other as the deformability of the bubbles is changed is examined. We show that as long as the bubbly wall-layer is maintained, the flow rate increases slowly with the bubble deformability, but once the bubbles are sufficiently deformable and leave the wall, there is a sharp increase in the flow rate.

[1]Research supported by DOE (CASL)

8:26AM A3.00003 A one-way coupled, Euler-Lagrangian simulation of bubble coalescence in a turbulent pipe flow[1] , MICHAEL MATTSON, Naval Surface Warfare Center, Carderock Division, KRISHNAN MAHESH, University of Minnesota — A bubble coalescence model is developed using an Euler-Lagrangian approach for unstructured grids. The Eulerian carrier fluid is solved using large-eddy simulation (LES) and the Lagrangian particle motion is solved using one-way coupled equations relating the turbulent motion of the carrier fluid to the forces on each discrete bubble. The collision process is deterministic; bubble-bubble collisions are assumed to be binary and are modeled using a hard-sphere approach. A stochastic approach is used to model coalescence, with the probability of coalescence being a function of the bubble-bubble interaction timescale and the time to drain fluid between the colliding bubbles. Coalescence in a bubbly, turbulent pipe flow without buoyancy is simulated with conditions similar to a microgravity experiment by Colin, Fabre and Dukler [Int. J. Multiphase Flow (1991) **17**:533–544] and excellent agreement of bubble size distribution is obtained between simulation and experiment. With increasing downstream distance, the number density of bubbles decreases due to coalescence and the average probability of coalescence decreases slightly due to an increase in overall bubble size.

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

8:39AM A3.00004 Bubble Fields in 3D (No Glasses Necessary) , JESSE BELDEN, Naval Undersea Warfare Center, SAI RAVELA, Massachusetts Institute of Technology, TADD TRUSCOTT, Brigham Young University, ALEXANDRA TECHET, Massachusetts Institute of Technology — Resolution of the three-dimensional (3D) bubble fields induced by a turbulent circular plunging jet is approached using 3D Synthetic Aperture Imaging (SAI). 3D SAI is ideally suited to investigate optically dense multiphase flows due to its ability to reconstruct volumes that contain partial occlusions. Instantaneous bubble sizes and locations in the plunging jet bubble fields are extracted from the volumes and presented for various jet heights. The data are compared with existing literature on bubble penetration depth and size distributions and show excellent agreement. A scaling law for total air concentration as a function of depth below the free-surface is proposed, and is believed to be the first presentation of such a result. Coupled with scaling laws for the maximum air concentration and radial concentration profiles, this new scaling law can be used to determine the entire air concentration profile given a minimal number of single point measurements.

8:52AM A3.00005 Shock propagation in a liquid containing bubbly clusters , HERVE GRANDJEAN, NICOLAS JACQUES, ENSTA Bretagne, STEPHANE ZALESKI, University Paris 06 — The propagation of shock waves in a liquid containing spherical bubbly clusters is investigated. A continuum model for the behaviour of such heterogeneous diphasic fluid has been developed, using a scale transition method. Numerical simulations of shock wave propagation reveal that the presence of bubble clusters has a significant influence on the shock dynamics. It is observed that the structure of the shock is dominated by the global response of the clusters instead of the single-bubble dynamics, as in homogeneous bubbly flow. As a consequence, the wavelength of the shock structure in a liquid with bubble clusters may be much larger than for the corresponding homogeneous bubbly liquid. To assess the accuracy of the proposed continuum modelling, axisymmetric simulations in which the position and shape of the clusters are directly specified were carried out. A good agreement between the two approaches was observed. The results of the proposed model were also found to compare well to experiments of the literature.

9:05AM A3.00006 A relation between the structures of a void fraction and liquid motion and the dissolved CO2 gas concentration, in a bubbly flow , MASAHIRO YAMADA, Graduate School Engineering, Shizuoka University, TAKAYUKI SAITO, Graduate School of Science and Technology, Shizuoka University — We discuss a relation between the structures of a void fraction and liquid motion and the CO_2 concentration, using a newly developed photoelectric optical fiber probe (POFP). The POFP is able to simultaneously measure a bubble diameter, velocity, time-series void fraction, and CO_2 concentration dissolved in the liquid around the bubbles. Moreover, we measured the velocity of the liquid phase at the same point, using LDV. We adopted a similar way of thinking into a consideration of the length scale to the liquid phase motions, void fraction and CO_2 concentration in order to evaluate spatial-scale. The integral-length-like scale of the liquid phase motions indicated few change against the height. The length scale of the void fraction became small toward the upper zone of the bubble column. On the other hand, the length scale of the CO_2 concentration exists up to the height of the middle zone. Based on these results, we discuss a relation of spatial-scale of the void fraction and the liquid motion to the CO_2 concentration.

9:18AM A3.00007 Two-Phase Flow Model and Experimental Validation for Bubble Augmented Waterjet Propulsion Nozzle[1] , J.-K. CHOI, C.-T. HSIAO, X. WU, S. SINGH, A. JAYAPRAKASH, G. CHAHINE, Dynaflow, Inc. — The concept of thrust augmentation through bubble injection into a waterjet has been the subject of many patents and publications over the past several decades, and there are simplified computational and experimental evidence of thrust increase. In this work, we present more rigorous numerical and experimental studies which aim at investigating two-phase water jet propulsion systems. The numerical model is based on a Lagrangian-Eulerian method, which considers the bubbly mixture flow both in the microscopic level where individual bubble dynamics are tracked and in the macroscopic level where bubbles are collectively described by the local void fraction of the mixture. DYNAFLOW's unsteady RANS solver, 3DYNAFS-VIS© is used to solve the macro level variable density mixture medium, and a fully unsteady two-way coupling between this and the bubble dynamics/tracking code 3DYNAFS-DSM© is utilized. Validation studies using measurements in a half 3-D experimental setup composed of divergent and convergent sections are presented. Visualization of the bubbles, PIV measurements of the flow, bubble size and behavior are observed, and the measured flow field data are used to validate the models. Thrust augmentation as high as 50% could be confirmed both by predictions and by experiments.

[1]This work was supported by the Office of Naval Research under the contract N00014-07-C-0427, monitored by Dr. Ki-Han Kim.

9:31AM A3.00008 Macroscopic phase-field model of partial wetting: bubbles in a capillary tube , LUIS CUETO-FELGUEROSO, RUBEN JUANES, Massachusetts Institute of Technology — Drops and bubbles are non-spreading, local, compactly supported features. They are also equilibrium configurations in partial wetting phenomena. Yet macroscopic theories of capillary-dominated flow are often unable to describe these systems. We propose a framework to model multiphase flow in porous media with non-spreading equilibrium solutions. We illustrate our approach with a one-dimensional model of two-phase flow in a capillary tube. Our model allows for the presence of compactons: non-spreading steady-state solutions in the absence of external forces. We show that local rate-dependency is not needed to explain globally rate-dependent displacement patterns, and interpret dynamic wetting transitions as the route from equilibrium, capillary-dominated configurations, towards viscous-dominated flow. Mathematically, these transitions are possible due to non-classical shock solutions and the role of bistability and higher-order terms in our model.

9:44AM A3.00009 Critical scaling in the rheology of damped random spring networks[1] , BRIAN TIGHE, TU Delft — Physical, biological, and engineered materials ranging from foams and emulsions to bioppolymer and bar-joint networks can be modelled as random networks of springs. We study the oscillatory rheology of random networks immersed in a viscous background fluid, and show how their response is intimately tied to the presence or absence of floppy modes in the zero frequency limit. The rheology displays dynamic critical scaling with three different regimes: viscous fluid, elastic solid, and shear thinning power law fluid. We give scaling arguments to explain all of the critical exponents and confirm our predictions with numerics.

[1]Supported by the Dutch Organization for Scientific Research (NWO)

9:57AM A3.00010 Pseudoturbulence and regime transition in polydisperse flow[1] , SANTOS MENDEZ-DIAZ, Universidad Autonoma de Nuevo Leon, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico, JOSE LUIS MUÑOZ-COBO, Universidad Politecnica de Valencia, SERGIO CHIVA, Universitat Jaume I, SIMON MARTINEZ, FAUSTO SANCHEZ, Universidad Autonoma de Nuevo Leon — Vertical polydisperse liquid-gas flow was experimentally studied to analyze the regime transition from bubbly to slug flows. Laser Doppler Anemometry (LDA) and multi-tip conductivity probes were used to measure local flow parameters as liquid velocity, void fraction and superficial gas velocity in a cylindrical pipe. Particle Reynolds number was ranged from 100 to 10000 and Weber number from near to 0 to 100. The void fraction was progressively increased to obtain bubbly, transition and slug flow regimes. The power spectral density obtained from liquid velocity shows a nearly constant decay energy exponent when radial void fraction distribution shows a wall-peak in bubbly regime, whereas in transition and slug regimes the exponent decay shows a non-constant value when void fraction increases. Critical flow conditions defined by Reynolds and Weber numbers are suggested to identify regime transition from bubbly to slug. Turbulence intensity and other local flow parameters are analyzed to validate the proposed criteria.

[1]Supported by PROMEP.

Sunday, November 20, 2011 8:00AM - 10:10AM –
Session A4 Drops I: Numerical Methods 307

8:00AM A4.00001 A new method for calculation of acting forces on a deformable droplet in shear flow[1] , YOUNGHO SUH, CHANGHOON LEE, Yonsei University — A numerical method for calculating drag and lift acting on a deformable droplet in linear shear flow is presented. In this study, a level set approach is adopted to handle deformation and break-off of the interfaces. In order to determine the acting force on a droplet in shear flow field, we adopt feedback forces which can maintain the droplet at a fixed position with efficient handling of deformation. The presented method is applied for numerical simulation of spherical, deformed, and oscillating droplets in uniform flow, and the numerical results are favorably compared with the data reported in the literature [Dandy and Leal, JFM (1989)], [Feng and Beard, J. Atmos. Sci. (1991)]. The computation demonstrates that the shape of droplet deforms from sphere to oblate ellipsoid by increasing the Reynolds and Weber numbers. For large inertial effects at high Reynolds number, the droplet eventually breaks up into smaller droplets. Based on the numerical results, drag and lift forces acting on a droplet are observed to strongly depend on the deformation. Also, the present method is proven to be applicable to a three- dimensional deformation of droplet in the shear flow, which cannot be properly analyzed by the previous studies.

[1]Authors acknowledge support from ERC program sponsored by National Research Foundation of Korea.

8:13AM A4.00002 Phase field model simulation of droplet deformation and breakup in wall bounded turbulence , LUCA SCARBOLO, DAFNE MOLIN, ALFREDO SOLDATI — Prediction of droplet breakup in turbulence is crucial in many industrial and environmental processes such as fluid mixing in stirred chemical reactors or fluid clusters dispersion in large scale oceanic currents. To this aim we analyze the deformation and breakup of a single droplet in turbulent channel flow. The fluids are considered incompressible, density-matched and viscosity-matched. We use a Phase Field Model (PFM) based on the Cahn-Hilliard/Navier-Stokes equations system. We simulated a wide range of Weber numbers (ratio between inertial forces and surface tension) spanning two orders of magnitude with a Reynolds number $Re = 100$ based on the friction velocity and on the channel half height. We validate the approach by comparing specific droplet parameters such as the average droplet deformation and the droplet deformation in time, against available experiments. Turbulent flow statistics are also computed to examine the energy exchanges between the droplet and the surrounding fluid.

8:26AM A4.00003 On a phase-field model for a miscible drop in a spinning drop tensiometer[1] , ANDREA BOGHI, ANATOLIY VOROBEV, University of Southampton — We examine shape transformations of a solute droplet immersed into a solvent-filled and sealed capillary tube subject to fast rotations around its axis, i.e. the configuration of the spinning drop tensiometer. Despite the fact, that a droplet is miscible, its dissolution occurs rather slowly, and under rotations a droplet becomes elongated, which is used to measure the dynamic surface tension of the solute/solvent interface. The Boussinesq approximation [1] of the full (quasi- compressible) Cahn-Hilliard-Navier-Stokes is used as a theoretical model to capture the droplet evolution. We found that the behaviour of a miscible droplet contained in a closed enclosure is strongly different from an immiscible one. Miscible droplets in general are thermodynamically unstable and ultimately dissolve, large partially miscible droplets however may remain stable with the size determined by the total mass balance. In the limit of high Prandtl numbers, droplet's shape changes quickly (on a convective time scale), so that quasi- stable droplets are observed with only weak hydrodynamic flows present. Such states remain thermodynamically unstable: droplets lose their mass and the droplet's interface properties changes on a long diffusive time scale.

[1] A. Vorobev, PRE 82, 056312 (2010).

[1]The work is supported by the EPSRC grant No. EP/G014337.

8:39AM A4.00004 Drops settling in sharp stratification with and without Marangoni effects[1] , FRANCOIS BLANCHETTE, AVI SHAPIRO, U.C. Merced — We present numerical simulations of drops settling in a layered ambient fluid. The ambient is made up of miscible fluids, with the top layer lighter than the lower one, representing fluid stratified through temperature or salinity variations. The surface tension between the ambient and the drop may be uniform or be smaller in the lower layer. Such a system is applicable to oil droplets settling or rising in the ocean. When surface tension is uniform, the drop slows down significantly as it encounters the transition region, due to entrained fluid from the upper layer, before accelerating again in the lower layer. When the lower surface tension is smaller, the drop suddenly accelerates through the transition region. We characterize these effects in terms of the sharpness of the transition, and the drop's Reynolds number.

[1]We acknowledge support from NSF grant DMS 0808129.

8:52AM A4.00005 Scaling in the Transition from Selective Withdrawal to Viscous Entrainment , CHRIS POMMER, MICHAEL HARRIS, OSMAN BASARAN, Purdue University — In selective withdrawal, fluid is withdrawn through a tube that has its tip suspended a distance S above a flat interface separating two fluids. When the withdrawal rate Q is low, the interface forms a steady-state hump and only the upper fluid is withdrawn. When Q is increased (or S decreased), the interface undergoes a topological transition so that the lower fluid is entrained with the upper one, forming a steady-state spout. Here, this discontinuous transition is analyzed computationally when both fluids are incompressible and Newtonian. The numerical method employed is an implicit method of lines ALE algorithm which uses finite elements with elliptic mesh generation. The new approach neither starts with a priori idealizations, as has been the case with previous computations, nor is limited to length scales above that set by the wavelength of visible light as in any experimental study. In particular, it is shown that the critical withdrawal rate at which the aforementioned transition occurs scales with the nozzle separation raised to some power n. A study of the effect of the physical parameters of the system on the scaling behavior is also presented.

9:05AM A4.00006 Interactions between two unilamellar vesicles , L. GARY LEAL, JOHANN WALTER, Department of Chemical Engineering, University of California, Santa Barbara — Suspensions of lipid vesicles are widely used in industrial applications such as personal and household care products. Interactions between vesicles can cause them to adhere and aggregate, which may dramatically modify the rheology of the suspension. In this work, we study the interactions of two unilamellar vesicles in a head-on collision. The study takes into account hydrodynamic phenomena, electrostatic repulsion between charged vesicles, depletion attraction forces due to polymers in solution and the effect of the deformation of the vesicles. Numerical simulations are conducted using an axisymmetric model coupling boundary integrals for the motion of the fluids and finite elements for the membrane mechanics. The results are compared with a new analytical scaling theory. Contrary to drops, it is shown that the drainage time is reduced when the driving force bringing the vesicles together increases. This is due to the increasing tension in the membrane as the vesicles get closer, which leads to a higher pressure in the film. It is also shown that the vesicles' ability to deform can significantly enhance the adhesion between them.

9:18AM A4.00007 Strategies for Efficient Microfiltration of Oil-in-Water Emulsions , TOHID DARVISHZADEH, NIKOLAI PRIEZJEV, Dept. of Mechanical Engineering, Michigan State University — This study addresses the issue of the separation of oil droplets from water for oil spill mitigation and produced water treatment. The effective separation of oil-in-water dispersions involves high flux of water through a membrane and, at the same time, high rejection rate of oil droplets, while avoiding membrane fouling. In this study, the effects of transmembrane pressure and crossflow velocity on rejection of oil droplets by pores of different cross-section are investigated numerically by solving the Navier-Stokes equation. We found that in the absence of crossflow, the critical transmembrane pressure, which is required for the oil droplet entry into a circular pore of given surface hydrophobicity, agrees well with analytical predictions based on the Young-Laplace equation. With increasing crossflow velocity, the shape of the oil droplet residing at the pore entrance is elongated along the flow and the critical pressure increases. In the case of pores with an elliptical cross-section, the water flux through the membrane is enhanced, in agreement with simple analytical considerations. The results of the numerical simulations are used to outline strategies for the experimental design of porous filters for oil spill remediation and produced water treatment applications.

9:31AM A4.00008 Interfacial Effects on Droplet Dynamics in Poiseuille Flow , JONATHAN SCHWALBE, MITRE, FREDERICK PHELAN JR., NIST, PETIA VLAHOVSKA, Brown University, STEVEN HUDSON, NIST — Many properties of emulsions arise from interfacial rheology, but a theoretical understanding of the effect of interfacial viscosities on droplet dynamics is lacking. Here we report such a theory, relating to isolated spherical drops in a Poiseuille flow. Stokes flow is assumed in the bulk phases, and a jump in hydrodynamic stress at the interface is balanced by Marangoni forces (linearized with respect to local deviations of interfacial surfactant concentration) and surface viscous forces according to the Boussinesq–Scriven constitutive law. Interfacial diffusion is also included. Our analysis predicts slip, cross-stream migration and droplet-circulation velocities. These results and the corresponding interfacial parameters are separable: e.g., cross-stream migration occurs only if gradients in surfactant concentration are present; slip velocity depends on viscosity contrast and dilatational properties, but not on shear Boussinesq number. This separability allows a new and advantageous means to measure surface viscous and elastic forces directly from the drop interface. Modeling of other geometries will also be discussed.

9:44AM A4.00009 Drops settling in a fluid with surface tension increasing with depth[1] , AVI SHAPIRO, FRANCOIS BLANCHETTE, U.C. Merced — We investigated numerically drops settling across layers of miscible fluids, representing oil droplets settling in a fluid stratified by temperature or salinity variations. The top layer is lighter than the lower one, while the drop itself is heavier than both layers. As the drop settles into the lower, its surface tension with the ambient fluid increases, which generates significant Marangoni effects. If the surface tension difference is small, the drop is delayed as it settles into the lower layer. Above a critical surface tension difference, the drop may be altogether prevented from crossing into the lower layer. We determine the conditions under which a drop may remain suspended at the transition region, and study the mixing generated by suspended drops.

[1]We acknowledge support from NSF grant DMS 0808129.

9:57AM A4.00010 A model for droplet condensational growth in a turbulent, axisymmetric jet , RYAN KEEDY, ALBERTO ALISEDA, University of Washington — Droplet growth at the edge of clouds is strongly influenced by the non-linear saturation field produced by mixing of warm, wet air inside the cloud with cold, dry air outside. This, together with the high intermittency of the turbulent at these geological scales, leads to uncertainty in the modeling of this process. We use experiments in a turbulent, axisymmetric jet to study this problem and develop a model. Although the distribution of a passive scalar in a turbulent jet is a classic problem, with a well-established solution, little attention has been devoted to heterogeneous nucleation, condensational growth and evaporation within a turbulent mixing layer where local supersaturation values may exceed unity. By leveraging the well-characterized self-similar behavior of a scalar (temperature, humidity) within a turbulent jet, we use a stochastic model for the instantaneous values from the statistics of the distribution to determine the super-saturation profile. Taking into account the high intermittency of the supersaturation field allows us to predict the droplet size at various stages of the flow. A Phase Doppler Particle Analyzer (PDPA) is used to collect statistics of velocity statistics, droplet growth and frequency that are used to inform the development and validation of the model.

Sunday, November 20, 2011 8:00AM - 9:44AM –
Session A5 CFD I: Turbulent Flow Simulations 308

8:00AM A5.00001 Direct numerical simulation of turbulent flow in a channel with different types of surface roughness[1] , IGOR A. BOLOTNOV, North Carolina State University — Direct numerical simulation (DNS) was performed for turbulent channel flow ($Re_\tau = 400$) for two types of wall surface roughness and well as smooth walls. The roughness elements of first type were assumed to be two-dimensional, transverse square rods positioned on both walls in a non-staggered arrangement. The height of the rods corresponds to $y^+ = 13.6$ and thus extends in the buffer layer. The second type of roughness was represented by a set of hemispherical obstacles (height of $y^+ = 10$) located on both channel walls and arranged on a square lattice. The presented simulations are part of benchmark problems defined by thermal-hydraulics focus area of the Consortium for Advanced Simulations of Light Water Reactors (CASL). This problem simulates the effect of the presence of growing bubbles on the walls of nuclear reactor fuel rods and aimed on evaluating CFD capabilities of various codes before applying them to more advanced problems. Mean turbulent quantities were computed and compared with available analytical and experimental results. The results of this work will be used to evaluate the performance of other LES and RANS codes on this benchmark problem.

[1]Supported by Consortium for Advanced Simulation of Light Water Reactors (CASL).

8:13AM A5.00002 ABSTRACT WITHDRAWN —

8:26AM A5.00003 Massively parallel LES of flow over a disc-golf frisbee , JOSHUA CAMP, Texas A&M University, YUVAL DORON, Exosent Engineering, ANDREW DUGGLEBY, Texas A&M University — Achieving industrially-relevant fluid simulations require reducing the total time from computer aided design (CAD) to simulation results. On the back-end, massively-parallel algorithms significantly reduce simulation time. On the front-end, automatic hexahedral meshing algorithms struggle to generate high-quality meshes, especially in near-wall regions where resolving the turbulent boundary layer is crucial to simulation accuracy. In this study, *a posteriori* mesh enhancements via boundary element insertion of an automatic hexahedral mesh is paired with a massively-parallel spectral element solver. This strategy leads to the potential for CAD to high-quality simulation results overnight. The method is tested in solving flow over a spinning disc-golf frisbee, which tests industrially relevant capabilities of a complex geometry under rotation. To solve, two numerical meshes are generated: the first mesh is coaxial with the frisbee where the flow is solved in the rotating frame of reference, and the second is external to the first in a stationary reference frame. The flow is solved using a massively parallel spectral element large eddy simulation algorithm using a high-pass-filtered based eddy viscosity. The near-perfect scaling for the present work provides the capabilities for reducing the total time from CAD to simulation results to under twelve hours, fast enough for Large Eddy Simulation (LES) and DNS to be used in a design-cycle.

8:39AM A5.00004 Assessment of Partially-Averaged Navier-Stokes method for prediction of fluid flow and heat transfer in a matrix of surface-mounted cubes , BRANISLAV BASARA, AVL List GmbH — The performance of the variable-resolution Partially-Averaged Navier-Stokes (PANS) method has been well documented for predictions of separated and wall bounded flows. However, its performance on applications which include a heat transfer is yet to be demonstrated. Therefore, the present work studies the flow in a matrix of surface-mounted cubes which is a well-known test case for the conjugate heat transfer. The recently proposed PANS z-f model is applied in conjunction with the hybrid wall treatment, which combines the integration up to the wall with wall functions. In adopted PANS approach, the filter width is controlled by specifying only one control parameter: an unresolved-to-total ratio of turbulent kinetic energy. The same modeling principles are applied on energy equation and on the wall heat transfer. Measurements, but also previous LES calculations, are used as a reference point to the present calculations.

8:52AM A5.00005 Distributed computation of the Kolmogorov-like cascades of the Couette flow by the arbitrary-precision differentiation , STANISLAV MIROSHNIKOV, Syncsort Inc. — The effect of the period of perturbations on the spatiotemporal statistics of the Kolmogorov-like cascades of the transitional Couette flow is explored using a new method of arbitrary-precision differentiation of trigonometric, hyperbolic, and elliptic structures. The trigonometric, hyperbolic, and elliptic structures are constructed and their differentiation is reduced to an algebraic processing, which may be executed with symbolic and numeric parameters. Computation of high-order derivatives by the arbitrary-precision differentiation and summation of the Boussinesq-Rayleigh-Taylor series for the perturbed Couette flow is implemented in Maple, Python, and C++. Performance of the proposed algorithms is compared both for workstations and clusters.

9:05AM A5.00006 Fluid-Structure Interaction Simulations of a Three-Dimensional Flexible Hydrofoil , BRENT CRAVEN, ROBERT CAMPBELL, Applied Research Laboratory, The Pennsylvania State University, YOUNG HWANG, THAD MICHAEL, SETH SCHROEDER, Naval Surface Warfare Center, Carderock Division — The deformation of a three-dimensional flexible hydrofoil was studied using two strongly-coupled partitioned fluid-structure interaction (FSI) approaches. Open-source (OpenFOAM), commercial (Abaqus), and custom software are coupled using two different mesh deformation and load/deflection interpolation methods. These approaches were used to carry out high-fidelity FSI simulations of the flow-induced deformation of the hydrofoil at various angles of attack. Large hybrid unstructured computational meshes, exceeding 20 million computational elements, were utilized to demonstrate the practical feasibility of the FSI approaches. Experimental validation of the computational results is presented that includes a comparison with particle image velocity (PIV) measurements of the flow field, force and moment data, and optical measurements of hydrofoil deflection.

9:18AM A5.00007 Numerical Simulation of Flows with Shocks and Turbulence using Observable Euler Equations , HARESHRAM NATARAJAN, KAMRAN MOHSENI, University of Florida — Problems involving shocks and turbulence in fluids are multi-scale in nature and are prone to continuous generation of high-wavenumber modes or small-scales. In practical applications, one is interested in an evolution equation for the large scale quantities without resolving the details of the small-scales. In this study observable Euler equations are employed in order to simulate flows with shocks and turbulence. Numerical results for 1D shock tube problem, 2D interaction of normal shock with vorticity/entropy wave, 3D Taylor Green problem will be reported and compared with the other available techniques.

9:31AM A5.00008 A Discontinuous Galerkin method for the compressible Navier-Stokes equations , SREENIVAS VARADAN, ERIC JOHNSEN, University of Michigan — Turbulence developed from instabilities such as Rayleigh-Taylor or Richtmyer-Meshkov poses a challenging problem for numerical simulations. While shock capturing is necessary when shocks are present in the domain, one also needs to minimize dissipation in the smooth parts of the solution so that the high wave numbers are not damped by numerical viscosity. Our numerical experiments with 3-D isotropic turbulence at high turbulent Mach numbers indicate that hybrid methods, which only use shock capturing selectively based on the dilatation sensor of Ducros et al, have a clear advantage in terms of computational cost and bandwidth resolution but cannot sense contact discontinuities and material interfaces. Motivated by this, we propose a hybrid method for the simulation of the Rayleigh-Taylor instability based on a new sensor that detects contacts and material interfaces based on characteristic variables. The sensor has been tested on 1-D as well as multi-dimensional problems. We use the shock-capturing ability of the 5^{th} order WENO and the 6^{th} order central flux away from discontinuities. We discuss the performance and merits of using this method as well as results for the Rayleigh-Taylor instability.

Sunday, November 20, 2011 8:00AM - 10:10AM – Session A6 Insect Flight I 309

8:00AM A6.00001 Impulse perturbation and recovery in a free flying insect[1] , TYSON HEDRICK, JEREMY GREETER, University of North Carolina at Chapel Hill — Flying animals are renowned for their ability to recover from substantial perturbations due to environmental turbulence, contact with other objects, or other sources. These capabilities are due to the underlying aerodynamics of flapping flight (its open-loop properties) and feedback control based on the neurosensory inputs, the combination providing the animal's closed-loop response. Here we examine the sources of perturbation resistance and recovery in a flying insect by delivering a pitch impulse perturbation to hovering hawkmoths (*Manduca sexta*) while characterizing the response of the animal using high-speed stereo videography. The closed-loop neurosensory and open-loop aerodynamic components of the response were then separated by comparing the observed response dynamics of the animal to those predicted from a variety of open loop models and simulations of hawkmoths. We found that the moth's perturbation response was dominated by its open-loop aerodynamic properties, which include sufficient damping to rapidly slow the pitch perturbation. A slower sensory response helped bring the moth back to a level orientation.

[1]Funded by NSF IOS-0920358 to TL Hedrick

8:13AM A6.00002 Improved Low-Order Models of Bio-inspired Pitching and Perching , CHENGJIE WANG, JEFF D. ELDREDGE, Mechanical & Aerospace Engineering, University of California, Los Angeles, CA, USA — To study the flying of small creatures, their simplest flapping motions, pitching and perching, are investigated by low-order inviscid point vortex models. These motions induce coherent vortex shedding at the leading edge, which has a profound influence on the generated force. Instead of fully recovering the flow field around wing, the reduced models track only small number of discrete vortices with time-varying strength to account for the unsteady aerodynamics. The idea of impulse matching is introduced to develop the new governing equation, different from the previously-developed Brown-Michael equation. For both pitching and perching motions, the results from the impulse matching model are compared with high fidelity simulations under different pitching rate and axis position, and this comparison shows a good qualitative agreement, which is better than obtained with the Brown-Michael approach. The results are also compared with previous experiments conducted in a water tunnel, and good qualitative agreement is achieved. Further, some detailed analysis of the high fidelity simulation has been performed to get intuition about the leading edge vortex, which can help us in improving the low-order model.

8:26AM A6.00003 Low Dimensional Modeling And Computational Analysis of Dragonfly Wing Aerodynamics[1] , YAN REN, HUI WAN, HAIBO DONG, Wright State University, FLOW SIMULATION RESEARCH GROUP TEAM — High-fidelity numerical simulations are being used to examine the key aerodynamic features and lift production of insect wings. However, the kinematics of the insect's wing and the resulting aerodynamics is highly complex, and does not lend itself easily to analysis based on simple notions of pitching/heaving kinematics or lift/drag based propulsive mechanisms. A more inventive approach is therefore needed to dissect the wing gait and gain insight into the remarkable aerodynamic performance of the insect's wing. The focus of the current investigation is on the aerodynamics of the wing of a dragonfly (Erythemis Simplicicollis) in hovering motion. The three-dimensional, time-dependent wing kinematics is obtained via a high-speed photogrammetry system. Singular Value Decomposition (SVD) is then applied to extract the essential features of the wing gait. The SVD spectrum shows that the first four modes capture more than 80% of the motion. Aerodynamics of wings flapping with kinematics synthesized from SVD modes will be discussed in detail.

[1]This work is supported by NSF CBET-1055949.

8:39AM A6.00004 Flight Forces Control Strategy in Insects[1] , SAMANE ZEYGHAMI, HAIBO DONG, Wright State University — Introducing a new point of view to steering muscles activity in insects, we are proposing a new flight forces control strategy that shows strong compatibility with many experimental observations. We have proven that the flight forces alternations can be controlled without direct control of wing's kinematic parameters. According to the proposed method choosing appropriate phase shift in steering muscles' activity between bilateral wings, insect is able to do desired maneuver. In fact correlation between the wing's motion and aerodynamic forces could give the insect high level of control on the flight forces magnitude and direction while requiring much simpler control system and lower power input.

[1]This work has been suppoterd by NFS, grant number CBET-1055949.

8:52AM A6.00005 Wing Damage Effects on Dragonfly's maneuverability[1] , ZHE NING, KUO GAI, SAMANE ZEYGHAMI, HAIBO DONG, Wright State University, FLOW SIMULATION RESEARCH GROUP (FSRG) TEAM — In this work, how the insect flight behavior contributes to its adaptability to limited performance condition is studied through a combined experimental and computational study. High speed photogrammetry is used to collect the data of dragonflies' takeoffs with intact and damaged wings along the chord and span separately. Then the effect of the spanwise and chordwise damage on the dragonfly wing is investigated. Results show that both changes have different effects on the wing and body kinematics and the merit of maneuverability. Two theories will be introduced to explain the wing damage tolerance behavior of the dragonfly flight.

[1]This work is supported by NSF CBET-1055949.

9:05AM A6.00006 Observation of the wing deformation and the CFD study of cicada[1] , HU DAI, SHAHRIZAN MOHD ADAM DAS, HAOXIANG LUO, Vanderbilt University — We studied the wing properties and kinematics of cicada when the 13-year species emerged in amazingly large numbers in middle Tennessee during May 2011. Using a high-speed camera, we recorded the wing motion of the insect and then reconstructed the three-dimensional wing kinematics using a video digitization software. Like many other insects, the deformation of the cicada wing is asymmetric between the downstroke and upstroke half cycles, and this particular deformation pattern would benefit production of the lift and propulsive forces. Both two-dimensional and three-dimensional CFD studies are carried out based on the reconstructed wing motion. The implication of the study on the role of the aerodynamic force in the wing deformation will be discussed.

[1]This work is sponsored by the NSF.

9:18AM A6.00007 ABSTRACT WITHDRAWN —

9:31AM A6.00008 Active Deformation in Insect Wings and Its Effect on Aerodynamic Performance of Flapping Flight[1] , LINGXIAO ZHENG, RAJAT MITTAL, TYSON HEDRICK — While passive deformation in insect wings during flight is well know, we demonstrate the presence of significant active deformation in the wings of butterfly. High-speed videography is used to capture wing kinematics of a Painted Lady butterfly flying freely inside an enclosure. An analysis of the videos shows that the chordwise deformation in the wings of this insect is inconsistent with passive (flow or inertial induced) deformation. It therefore follows that the deformation is produced actively by the insect. Computational fluid dynamics analysis is used to examine the effect of this active deformation on aerodynamic performance and results from this study will be presented.

[1]Research is supported by NSF and AFOSR

9:44AM A6.00009 Understanding Flight Stabilization In Insects to Large Perturbations Using an Integrated Experimental-Numerical Approach[1] , CHAO ZHANG, LINGXIAO ZHENG, The Johns Hopkins University, TY HEDRICK, The University of North Carolina, RAJAT MITTAL, The Johns Hopkins University — Insects adopt a variety of strategies to stabilize flight in the face of large-scale aerodynamic and mechanical perturbations. Linear stability analysis and simple estimation techniques such as blade-element models can only provide very limited insights into these strategies since the response is highly non-linear and far from simple equilibrium conditions. In order to tackle this problem, we have coupled a Navier-Stokes immersed boundary-based solver to a six degree-of-freedom (DOF) flight dynamics model of a Hawkmoth (Manduca Sexta) to model this stabilization process in all its complexity. The model is used to examine the response of this insect to large-scale aerodynamic and mechanical perturbations and results from this study will be presented.

[1]This research is supported by AFOSR.

9:57AM A6.00010 Understanding the Role of Moment-of-Inertia Variation in Insect Flight Maneuvers[1] , TIRAS LIN, RAJAT MITTAL, LINGXIAO ZHENG, The Johns Hopkins University, TYSON HEDRICK, The University of North Carolina at Chapel Hill — The objective of this study is to gain insights into insect flight maneuvers and, in particular, the role that changes in body moment-of-inertia might play during these maneuvers. High-speed, high-resolution videogrammetry is used to quantify the trajectory and body conformation of Painted Lady butterflies during flight maneuvers; the 3D kinematics of the center-of-masses of the various body parts of the insect are determined experimentally. Measurements of the mass properties of the insect are then made and used to parameterize a simple flight dynamics model of the butterfly. Even though the mass of the flapping wings is small compared to the total mass of the insect, these experiments and subsequent analyses indicate that changes in moment-of-inertia during flight are large enough to have a noticeable impact on the maneuvers of these insects.

[1]Research is supported by NSF and AFOSR.

Sunday, November 20, 2011 8:00AM - 10:10AM —
Session A7 Turbulent Boundary Layers I 310

8:00AM A7.00001 Predictive wall model and LES applied to the flat-plate turbulent boundary layer , M. INOUE, California Institute of Technology, R. MATHIS, I. MARUSIC, University of Melbourne, D.I. PULLIN, California Institute of Technology — An empirical inner-outer wall model (Mathis *et al*, *JFM 2011*) is used, together with time series of stream-wise, resolved-scale velocities within the logarithmic region obtained from large-eddy simulations (LES), to calculate turbulence intensities $\overline{u'^2}/u_\tau^2$ in the inner region of the zero-pressure gradient turbulent boundary layer. Comparisons are made of the LES-wall-model results with both equivalent predictions using experimental time series, and also with direct experimental measurements at $Re_\tau = 7,300, 13,600$ and $19,000$. LES combined with the wall model are then used to extend the inner-layer predictions to Reynolds numbers within a gap in $\log(Re_\tau)$ space between laboratory measurements and surface-layer, atmospheric experiments.

8:13AM A7.00002 Low-order representation of turbulent pipe flow[1] , JEAN-LOUP BOURGUIGNON, BEVERLEY MCKEON, California Institute of Technology — A simple model of pipe flow turbulence based on a forcing-response analysis of the Navier-Stokes equations (McKeon *et al.* 2010) is presented and used to investigate the dynamics of the large-scale structures. The model is validated against pipe flow DNS data and naturally leads to a low-order representation of the flow as a sum of traveling waves corresponding to the response modes predicted by the model. The low-order flow representation captures a significant fraction of the turbulence intensity and Reynolds stress in the core of the pipe and is highly correlated to the large-scale structures observed in the DNS data. Furthermore, by considering the forcing modes associated with each response mode in the low-order flow representation the amplification mechanisms sustaining the large-scale structures can be identified.

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

8:26AM A7.00003 Classification of critical points in a converging-diverging turbulent channel flow

, RAÚL BAYOÁN CAL, Portland State University, MURAT TUTKUN, ØYVIND WAAGE HANSSEN-BAUER, ANDERS HELGELAND, ØYVIND ANDREASSEN, Norwegian Defence Research Establishment, JEAN-PHILIPPE LAVAL, Laboratoire de Mecanique de Lille — Based on Direct Numerical Simulations of a channel with a bump,[1] a critical point generation and classification is performed. Topological features of the flow are found relating critical point type with the distribution of Jacobian Determinants as well as the behavior of skin friction. Eight different types of points are found within the flow and these are visualized using different forms of the Line Integral Convolution visualization technique as described by Aasen and Furuheim (2008).[2] It is found that the determinant analysis can be a useful method for data reduction of critical points. Local minima and maxima of the skin friction coefficient, C_f, are reliable indicators of development of extreme determinants. Further explanation of the physics is done through the analysis of the velocity and pressure iso-surfaces.

[1]M. Marquillie et al. (2008), J. Turbulence, vol 9, no 1, pp. 1-23.

[2]M. Aasen and K. Furuheim (2008), M.Sc. thesis, Department of Informatics, University of Oslo, Oslo, Norway.

8:39AM A7.00004 Spatial locality of turbulent fluxes: the filtering approach

, SCOTT SALESKY, MARCELO CHAMECKI, Pennsylvania State University — A number of methods exist for decomposing turbulent fluxes, including those that are local in scale (e.g. cospectra, structure functions) or in scale and space (e.g. wavelets). We propose a new spatially local decomposition of turbulent fluxes based on a filtering operation where the filter width Δ defines the scale of the local fluctuations. The Germano identity is used to show the ensemble average of the local flux recovers the Reynolds-averaged flux at any given scale Δ. Properties of local flux-gradient relationships are investigated using atmospheric surface layer data from the Advection Horizontal Array Turbulence Study (AHATS). In atmospheric surface layer measurements, random error in measured turbulent fluxes occurs due to insufficient averaging times for the time mean to converge to the ensemble mean by the ergodic hypothesis. Maximum possible averaging times are limited due to nonstationary large scale atmospheric motions. A new method of estimating random error in atmospheric surface layer data based on local fluxes is evaluated, and other applications of local fluxes are considered.

8:52AM A7.00005 Turbulent Boundary Layers: An Energy Harvesting Perspective[1]

, PIERRE LEMAIRE, INSA de Lyon, France, HUSEYIN DOGUS AKAYDIN, NIELL ELVIN, YIANNIS ANREOPOULOS, The City College of New York — A turbulent boundary layer (TBL) carries mechanical energy distributed over a range of temporal and spatial scales. The inherent unsteadiness in the TBL induces a strain field on a solid body immersed in it. The induced strain can be converted to electrical energy using a solid body of piezoelectric material. This energy harvesting method can be used for developing self-powered flow sensors. In the present work, we experimentally investigate the interaction of a TBL with a thin flexible beam. The vibration frequency and amplitude of the beam is measured using strain gages. Three relevant parameters are the length of the beam (l), the distance of the beam from the wall (h) and the free stream speed (V_∞). While V_∞ changes the TBL characteristics, h and l primarily affect the fluid-structure interaction. In our wind tunnel tests we traversed the piezoelectric beam across the TBL by varying these three parameters for the purpose of finding values maximizing the vibrations. We present a "power map" of the TBL indicating the optimal h and V_∞ values for a given value of l. We also discuss the effect of l in flow-induced vibrations by presenting spectrum analysis of strain signals at various h and V_∞.

[1]Sponsored by NSF Grant: CBET #1033117.

9:05AM A7.00006 A Multi-scale Approach for the Thermally Stratified Turbulent Boundary Layer[1]

, SAM NOTARO, Syracuse University, GUSTAVO RIVERA-ROSARIO, Rensselaer Polytechnic Institute, LUCIANO CASTILLO, Texas Tech University — The governing equations have been established for the inner and outer regions of the turbulent boundary layer for thermally stratified flows subject to external pressure gradient. It was shown that the pressure gradient across the boundary layers contains the wall-normal Reynolds stress and the buoyancy term. Therefore, the streamwise momentum equation includes the buoyancy terms and can be employed to obtain the scales of the velocity and thermal fields. Furthermore, it has been clearly identified that single scaling analysis cannot satisfy the equations of motion while satisfying the boundary layers simplifications. Moreover, using a multi-scale similarity approach of the equations of motion a buoyancy parameter, β_T has been derived and must be constant for an equilibrium flow to exist. From this new parameter a power law for the growth of the thermal boundary layer exists given as, $\delta_T \sim {\Delta T_w}^{1/\beta_T}$, in which the growth is controlled by the strength of the stratification. In addition, it was shown that the outer mean temperature scales with the temperature difference, ΔT_w, and the inverse of the Richardson number.

[1]NSF-CBET and ONR (R. Joslin)

9:18AM A7.00007 Linear analysis of transient growth in stably-stratified, turbulent channel flow[1]

, JUAN CARLOS DEL ALAMO, MAE Department, UC San Diego, CARLOS YANEZ, School of Aeronautics, Universidad Politecnica de Madrid, MANUEL GARCIA-VILLALBA, Dept. of Aerospace Engineering and Bioengineering, U Carlos III Madrid — We studied stably-stratified, fully-developed, turbulent channel flow using linear stability analysis. The analysis considered the mean velocity and density profiles extracted from DNS [1] calculations as base flow, and included a linear model to represent the energy dissipation and scalar diffusion felt at the large scales as a consequence of the small scales. The flow was found to be asymptotically stable in all cases but transient growth of initial perturbations was observed. The perturbations showing maximal transient growth corresponded well with spanwise waves in the center of the channel and with streaks in the near wall region, which were both observed in the DNS. In particular, their sizes and convection velocities were reasonably well predicted by the linear model. Component-wise analysis revealed that, while the streaks were formed by the vertical stirring of mean shear, the transient amplification of the spanwise waves was reminiscent of the Orr mechanism.

[1] M. García-Villalba and Juan C. Del Álamo, Turbulence modification by stable stratification in channel flow. Phys. Fluids, 23:045104, 2011.

[1]Supporte by the German Research Foundation, Project GA 1360/2-1.

9:31AM A7.00008 Inverse Magnus effect on a rotating sphere[1]

, JOOHA KIM, HYUNGMIN PARK, HAECHEON CHOI, JUNG YUL YOO, Seoul National University — In this study, we investigate the flow characteristics of rotating spheres in the subcritical Reynolds number (Re) regime by measuring the drag and lift forces on the sphere and the two-dimensional velocity in the wake. The experiment is conducted in a wind tunnel at $Re = 0.6 \times 10^5 - 2.6 \times 10^5$ and the spin ratio (ratio of surface velocity to the free-stream velocity) of 0 (no spin) - 0.5. The drag coefficient on a stationary sphere remains nearly constant at around 0.52. However, the magnitude of lift coefficient is nearly zero at $Re < 2.0 \times 10^5$, but rapidly increases to 0.3 and then remains constant with further increasing Reynolds number. On the other hand, with rotation, the lift coefficient shows negative values, called inverse Magnus effect, depending on the magnitudes of the Reynolds number and spin ratio. The velocity field measured from a particle image velocimetry (PIV) indicates that non-zero lift coefficient on a stationary sphere at $Re > 2.0 \times 10^5$ results from the asymmetry of separation line, whereas the inverse Magnus effect for the rotating sphere results from the differences in the boundary-layer growth and separation along the upper and lower sphere surfaces.

[1]Supported by the WCU, Converging Research Center and Priority Research Centers Program, NRF, MEST, Korea.

9:44AM A7.00009 A novel Lie-group analysis for wall-bounded turbulent flows , XI CHEN, ZHEN-SU SHE, Peking University, FAZLE HUSSAIN, University of Houston, SHE TEAM — Symmetry analysis based on the Lie-group method is the most effective method for solving nonlinear problems. Here, we present a novel Lie-group analysis for wall-bounded turbulent flow in following ways: First, the governing equation for symmetry analysis is inner and outer mean momentum equation, instead of the Navier-Stokes equation. Second, the dilation and Galilean transformations are applied to space variable, mean velocity, and in particular to the mixing length and its spatial gradient. Finally, a transition ansatz is formulated, as a special choice of the similarity solution, which accomplishes a composite solution across adjacent layers. With all of these, we achieve a rare occasion that symmetry is used to construct the complete solution: an analytic expression for the entire mixing length profile and then the mean velocity profile. Thus, a classical turbulence closure problem is analytically solved by combining multi-layer perturbation with Lie-group analysis.

9:57AM A7.00010 Accurate determination of Karman constant and mean velocity in high Reynolds number turbulent pipe , ZHEN-SU SHE, XI CHEN, YOU WU, Peking University, FAZLE HUSSAIN, University of Houston, SHE TEAM — In 1904, Prandtl1 presented the boundary layer concept, which initiated a century old search for analytic solution of turbulent flow near a wall. Here, we present a theory combining multi-layer perturbation with Lie-group analysis, for a complete determination of the similarity solution in turbulent channel and pipe flows. In particular, we present a procedure enabling close interaction with empirical data, which accomplishes an objective determination of key parameters involved in the solution. A remarkable outcome is all data suggest a universal Karman constant, having the value of 0.45. Based on measured parameter, the predicted MVPs agree with experimental/simulation data within 1% at all points for a wide range of Re covering nearly three decades. Because both multi-layer and Lie-group symmetry are general, this is a promising framework for mean flow solutions of a wide class of turbulent flows, including incompressible, compressible and rough-wall boundary layers, and Rayleigh-Benard convection.

Sunday, November 20, 2011 8:00AM - 9:57AM –
Session A11 Turbulence Theory I 314

8:00AM A11.00001 Can turbulence closure models come with error bars or uncertainty bounds? , SHARATH GIRIMAJI, Texas A&M University — The inherent complexity of turbulence – chaotic and intermittent character – may render many of the conventional uncertainty quantification norms and procedures ineffective for error assessment in closure models. A concerted alliance between turbulence physics and statistical characterization is imperative to make fundamental progress. In this presentation we first identify the most important sources of uncertainties in turbulence closure modeling and classify them into aleatoric (statistical) and epistemic (systematic) categories in the context of one-point closures. Then we propose a framework for exploiting the closure modeling knowledgebase accumulated over the last several decades and channeling the physical understanding for the purpose of uncertainty quantification. In the long run, such development is critical for the progression of turbulence CFD calculation from "post-dictive" nature to high-confidence predictive tools.

8:13AM A11.00002 A Study of Structures with Intense Vorticity in Isotropic Turbulence , ANTHONY LEONARD, California Institute of Technology — We study the characteristics of vortex structures having vorticity occupying the high amplitude tail of the distribution of vorticity amplitudes in homogeneous, isotropic turbulence. The data are obtained from the results of a 1024^3 DNS at $Re_\lambda = 433$ residing in the Johns Hopkins web-based public database (http/turbulence.pha.jhu.edu). The power law for the observed tail implies a certain average local structure for those regions containing intense vorticity. To obtain a global structure satisfy the solenoidal condition for the vorticity field we add the gradient of a vorticity potential computed using the above-mentioned local structure. Then by assuming that the full vorticity field results from a Poisson distribution in space of these global structures, we can then compute the full PDF of vorticity amplitudes employing techniques used to find the (Holtsmark) distribution of gravitational forces acting on a star due to neighboring stars (see, e.g., S. Chandrasekar, 1943, Rev. Mod. Phys. **15**, 1-89). The computed full PDF agrees very well with the observed PDF derived from the DNS data.

8:26AM A11.00003 A Lagrangian view of vorticity-strain alignment in turbulent flows , EBERHARD BODENSCHATZ, MPI-DS, Goettingen, ALAIN PUMIR, ENS Lyon, HAITAO XU, MPI-DS, Goettingen — Vortex stretching is arguably the most important aspect of dynamics in 3- dimensional fluid turbulence. Intuitively, one would expect that the vorticity vector is aligned to the direction of the strongest stretching. It is therefore puzzling when numerical simulations and experiments showed that, at any instantaneous time, vorticity preferentially aligns to the intermediate eigenvalues of the rate of strain tensor, corresponding to much weak stretching. Here we show that the dynamics is simplified when the alignment process is studied in a Lagrangian frame, i.e, following a fluid element. Using data from particle tracking experiments and direct numerical simulations, we studied the evolution of 4 fluid tracers that initially form isotropic tetrahedra. The evolution dynamics can be revealed in terms of the velocity gradient tensor perceived by the tetrads. For tetrads with size spanning from the dissipative scale to well in the inertial range, we observed the alignment of the vorticity to the direction of the strongest stretching *at earlier time*, i.e., a demonstration of the expected vortex-stretching in turbulence. For tetrads of sizes within the inertial range, the alignment process occurs at time scales given by the tetrad size and the energy dissipation rate.

8:39AM A11.00004 Extracting Coherent Structures from Turbulence[1] , GARY CHANDLER, RICH KERSWELL, University of Bristol — We consider the problem of extracting dynamically-important exact solutions of the Navier-Stokes equations directly from turbulent flows. By monitoring near-recurrences of the flow in direct numerical simulations of 2D body-forced turbulence, we uncover an array of equilibrium points, travelling waves and periodic orbits over a range of Reynolds numbers, which underpin the complicated dynamics seen. Progress will be discussed in determining whether these solutions can then be used to predict the mean statistics of the turbulence (in the spirit of Kawahara & Kida 2001).

[1]Supported by the EPSRC.

8:52AM A11.00005 Wavelet versus Fourier Analysis of the Conditional Vorticity Budget in Homogeneous Isotropic Turbulence , BENJAMIN KADOCH, Aix-Marseille Universite & M2P2–CNRS Ecole Centrale de Marseille, MICHAEL WILCZEK, Institute for Theoretical Physics, University of Muenster, KAI SCHNEIDER, M2P2–CNRS & CMI, Universite de Provence, RUDOLF FRIEDRICH, Institute for Theoretical Physics, University of Muenster, MARIE FARGE, LMD–CNRS, Ecole Normale Superieure — We study the conditional balance of vortex stretching and vorticity diffusion of fully developed three-dimensional homogeneous isotropic turbulence with respect to coherent and incoherent flow contributions. This decomposition is achieved by the Coherent Vorticity Extraction (CVE) based on orthogonal wavelets applied to DNS data, which yields insights into the influence of the different contributions as well as their interaction. It is shown that CVE yields an excellent representation of the total flow using a reduced number of degrees of freedom, which is particularly interesting as the conditional budget of vortex stretching and vorticity diffusion represents a dynamical rather than a purely kinematic relation. The results are compared to a decomposition with a standard Fourier filter.

9:05AM A11.00006 Convergence of third-order velocity structure functions in axisymmetric turbulence , FABIEN GODEFERD, LMFA UMR 5509 CNRS, Ecole Centrale de Lyon, ALEXANDRE DELACHE, LMFA Université Jean Monnet, Saint-Etienne — Kolmogorov theory (1941) for isotropic turbulence establishes asymptotic scaling laws for the statistics of n-th order structure functions at high Reynolds number, in terms of dissipation ϵ and separation distance r for the velocity increment δu. A famed relationship is the -4/5 law. When the turbulent flow is anisotropic, due to external distortions (background rotation,...) to inhomogeneities or initial conditions (jets, "isotropic" grid turbulence), such laws may fail. We examine the applicability of the K41 predictions for third-order moments of velocity structure functions, and evaluate low Reynolds number effects and anisotropic effects on the departure with the -4/5 law. We consider rotating or stably stratified turbulence, whose statistics are obtained by Direct Numerical Simulations or by a two-point statistical model allowing to reach high Reynolds numbers. We link anisotropic spectral statistics for energy transfer with $< (\delta u)^3 >$ and derive physical space statistics from spectral data of the statistical model. Although K41 scalings may arguably not apply to anisotropic turbulence, some justifications for anisotropic turbulence statistics can be provided (Taylor et al. PRE 2003) by specific data processing in DNS.

9:18AM A11.00007 Does the Kolmogorov scaling bridge hydrodynamic linear stability and turbulence? , STEFANIA SCARSOGLIO, FRANCESCA DE SANTI, DANIELA TORDELLA, Politecnico di Torino — The way in which the kinetic energy is distributed over the multiplicity of scales is a fundamental feature of turbulence. According to Kolmogorov's 1941 theory, by dimensional analysis, the only possible form for the energy spectrum function is the -5/3 spectrum. Experimental evidence has accumulated that supports it. Until now, such a power-law decay was considered a specific treat of the nonlinear interaction overlooking turbulence dynamics. We show here that this picture is also present in the linear interaction relevant to three-dimensional stable perturbation waves. Through extensive computation of the transient life of these waves in typical shear flows, we observe that the energy they own when, out of the transient phase, enter the final exponential decay shows a spectrum very close to the -5/3 spectrum. Moreover, also the observation times show a similar scaling.

9:31AM A11.00008 Statistically non-stationary turbulence[1] , WILLIAM K. GEORGE, Imperial College of Science, Technology and Medicine — Of all the assumptions in Kolmogorov 1941 the most fundamental is the assumption (or postulate) that that the smallest scales of the turbulence are in statistical equilibrium. This is usually justified by heuristic arguments based on the decreasing time scales as the eddy size is decreased relative to that of the energy containing eddies so that they can be assumed to be in *local equilibrium*. This argument can be shown to be fundamentally flawed since it does not account for the decreasing energy of the smaller scales. Moreover there are well-documented counter-examples to local equilibrium in non-stationary flows. It will be suggested that the K41 scaling arguments (or variations upon it) for the dissipative scales are in fact only applicable to flows in strict statistical equilibrium (i.e., statistically stationary), and should not be expected to apply to non-equilibrium flows. At least three types of non-equilibrium flows will be identified, one of which curiously enough satisfies Kolmogorov scaling at all scales.

[1] Supported by the Marie Curie Intra European Fellowship program.

9:44AM A11.00009 Restricted Euler moments in a pre-turbulent state , ROBERT M. KERR, ANDREW N. FERGUSON, University of Warwick, MIGUEL D. BUSTAMANTE, University College Dublin — The evolution of the velocity stress moments of the 'restricted Euler' equations with an isotropic pressure Hessian are compared with their evolution under the three-dimensional Hessian of the full nonlinear terms for a flow with changing structures. Past analysis of numerical data in a fully developed flows has shown that the second and third invariants, Q and R are distributed in the manner predicted by this model. In this work, we ask how these distributions depend upon whether the underlying vortex structures are sheets or tubes. Distributions will also be used to compare the time-derivatives of the individual terms, as predicted by the restricted Euler model, with their values when the full Hessian is applied. Both types of distributions are time-dependent, that is dependent on the underlying vortical structures. When vortex tubes dominate and the turbulence is becoming fully-developed, the predictions of the restricted Euler model are approximately obtained. However, when vortex sheets are dominant, another paradigm is needed. At a minimum, these results suggest that while the restricted Euler model might be a good representation of turbulence once it is fully developed, during the period when the turbulence is still developing, modifications to the usual picture will be needed.

Sunday, November 20, 2011 8:00AM - 10:10AM –
Session A12 Turbulence Simulation I 315

8:00AM A12.00001 Alternative Forcing for Homogeneous Isotropic Turbulence in Real Space , G. MALLOUPPAS, B.G.M. VAN WACHEM, W.K. GEORGE, Mechanical Engineering, Imperial College London, Exhibition Road, London SW7 2AZ — An alternative to the linear forcing of Lundgren [1] is suggested for sustaining homogeneous and isotropic turbulence. The method depends on a random pseudo-velocity field produced initially from an arbitrary spectrum. Energy can be fed into a variety of different range of wavenumbers so that the resulting total turbulent kinetic energy remains constant. This talk will first introduce the methodology used to obtain 1D spectra and correlation functions which will then be used to validate the homogeneity and isotropy of the sustained turbulence. The effects of window functions on 1D spectra and correlations will be also addressed. Finally, the effect of initial conditions (i.e. triggering at different wavenumbers) on the statistics, especially spectra and correlation functions will be investigated, with a particular view toward understanding the variety of spectra generated by experiments, e.g., [2,3].

[1] Lundgren, T. S. (2003) Ann. Res. Briefs, CTR, Stanford
[2] Comte-Bellot, G. and Corrsin, S. (1971) JFM, 48, 273-337.
[3] Seoud, R. and Vassilicos, J. C. (2007) Phys Flds. 19.

8:13AM A12.00002 ABSTRACT WITHDRAWN —

8:26AM A12.00003 Methods and issues for highly-scalable simulation of isotropic homogeneous turbulence , ORLANDO AYALA, CHARLES ANDERSEN, HOSSEIN PARISHANI, LIAN-PING WANG, University of Delaware — Direct numerical simulations (DNS) of homogeneous isotropic turbulence have served as a reliable quantitative research tool for studying the physics and dynamics of small-scale turbulence. It is desirable to extend the range of scales or equivalently the flow Reynolds number so a more complete understanding of related physical processes can be obtained. While the pseudo-spectral method is known to be the most accurate DNS approach, it has the challenge of performing global data communication (i.e., Fast Fourier Transform) which may not scale well on the state-of-the-art PetaScale computers with O(100,000) processors. Alternative approaches, such as high-order finite difference, finite-volume, and lattice Boltzmann equation, have been proposed to replace the pseudo-spectral method. The question we wish to address in this talk is: how do different approaches compare in terms of accuracy and parallel efficiency? Complexity analyses of several approaches will be discussed and be used to understand scalability data obtained from several parallel computers. The simulated flows will also be compared to examine the resolution requirements for various approaches to achieve a comparable accuracy. Detailed implementation issues related to multi-dimensional domain decompositions and large-scale forcing scheme will be discussed.

8:39AM A12.00004 A comparison between eddy-viscosity models and direct numerical simulation: the response of turbulent flow to a volume force

PAOLO LUCHINI, SERENA RUSSO, DIIN – Università di Salerno – Italy — Inspired by P. Luchini & F. Charru's[1] analysis of the phase lead of the wall-shear stress at a channel's perturbed bottom, we identified a benchmark problem simple enough that it can be solved both by an eddy-viscosity model, similar to those typically used in shallow-water flow calculations, and by direct numerical simulation. This is the linear response of a turbulent flow's mean-velocity profile to an external volume force. Such a force, of unspecified origin in the present context, was found in[2] to be representative of the perturbation induced by bottom topography, and its consequences were analysed by means of an eddy-viscosity model. On the other hand, a modification of Luchini, Quadrio & Zuccher's[3] method to compute the linear impulse response of a wall-bounded turbulent flow allows the response to a volume force to be computed directly. The comparison exhibits significant differences and suggests that there might be fundamental obstacles to designing an eddy-viscosity model that provides the correct result.

[1]Luchini, P., Charru, F., The phase lead of shear stress in shallow-water flow over a perturbed bottom, *J. Fluid Mech.* **665**, 516-539 (2010)
[2]Ibid.
[3]Luchini, P., Quadrio, M., Zuccher, S., The phase-locked mean impulse response of a turbulent channel flow, *Phys. Fluids* **18**, 121702 (2006).

8:52AM A12.00005 ABSTRACT WITHDRAWN —

9:05AM A12.00006 DNS of very strong adverse pressure gradient flows with eventual separation

GUILLERMO ARAYA, LUCIANO CASTILLO, Dept. of Mechanical Eng., Texas Tech University, Lubbock, TX 79409 USA — Direct Numerical Simulations (DNS) of spatially-evolving turbulent boundary layers with prescribed very strong adverse pressure gradients with eventual separation are performed. The driven force behind this investigation is to analyze the interaction between the inner and outer layers in separated flows. Also, the outer peaks in velocity fluctuations are analized by means of the energy budget of the turbulent kinetic energy and shear Reynolds stresses. A method for prescribing realistic turbulent velocity inflow boundary conditions is employed and based on the rescaling-recycling method proposed by Lund et al. (1998). The standard rescaling process requires prior knowledge about how the appropriate velocity and length scales are related between the inlet and recycle stations (e.g. classic scaling laws). Here a dynamic approach is proposed in which such information is deduced dynamically by involving an additional plane, the so called "test plane" located between the inlet and recycle stations. The approach also distinguishes between the inner and outer regions of the boundary layer and enables the use of multiple velocity scales, Araya et al. (2009, 2011). This flexibility allows applications to boundary layer flows with arbitrary pressure gradients.

9:18AM A12.00007 Slow Growth Formulation for DNS of Temporally Evolving Boundary Layers

VICTOR TOPALIAN, ONKAR SAHNI[1], TODD OLIVER, ROBERT MOSER, University of Texas at Austin — A formulation for DNS of temporally evolving boundary layers is developed and demonstrated. The formulation relies on a multiscale approach to account separately for the slow time evolution of statistical averages, and the fast time evolution of turbulent fluctuations. The source terms that arise from the multiscale analysis are modeled assuming a self-similar evolution of the averages. The performance of the formulation is evaluated using DNS of spatially evolving compressible boundary layers. This formulation was developed to provide data for the calibration of turbulence model parameters and enable the quantification of uncertainty due to the models. The extension of this formulation to homogenize spatially evolving boundary layers will also be discussed. This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

[1]Current Affiliation: Rensselaer Polytechnic Institute

9:31AM A12.00008 ABSTRACT WITHDRAWN —

9:44AM A12.00009 ABSTRACT WITHDRAWN —

9:57AM A12.00010 Dynamics of large-scale coherent structures in developing and fully developed channel flow over a rough wall

KYOUNGSIK CHANG, Hanseo University, GEORGE CONSTANTINESCU, University of Iowa — LES of fully developed and developing channel flow are conducted to study the formation and structure of the streaks for the case when the rough bed consists of an array of identical sinusoidal waves with an amplitude of 0.1λ, where λ is the wave wavelength. The channel depth is λ, its width is 10λ and the channel Reynolds number is 6,700. Consistent with experimental investigations, the average spanwise spacing of the streaks predicted by LES is close to 1.5λ in the case of a fully developed flow. A developing channel flow simulation with steady inflow conditions reveals the streaks form as a result of the break up of the quasi two-dimensional spanwise-oriented eddies forming in the shear layers past the crests of the first waves into arrays of pronounced hairpin vortices. These hairpins scale with the size of the waves which is of the order of the channel depth. Streaks are observed starting in the region where the flow transitions to turbulent. In average, the streaks become larger as one moves downstream until fully developed conditions are reached.

Sunday, November 20, 2011 8:00AM - 10:10AM —

Session A13 Biofluids: Cardiovascular: Pathological Flow and Intervention 316

8:00AM A13.00001 Virtual Cardiac Surgery Using CFD: Application to Septal Myectomy in Obstructive Hypertrophic Cardiomyopathy[1]

VIJAY VEDULA, RAJAT MITTAL, THEODORE ABRAHAM, The Johns Hopkins University — Obstructive hypertrophic cardiomyopathy (HOCM) is characterized by ventricular wall thickening, diastolic dysfunction, and dynamic outflow tract obstruction, all of which strongly influence the vortex dynamics and pressure distribution in the left ventricle (LV). Severe cases of HCM are usually managed through septal myectomy where the surgeon resects the hypertrophic mass. Surgeons currently try to remove as much tissue as possible in order to optimize the post surgical result. However, excessive debulking increases the chance of ventricular septal defects, bundle branch block or complete heart block, and aneurysmal septal thinning. On the other hand, insufficient tissue removal also leads to unsatisfactory outcomes in terms of reduction of outflow tract pressure gradient. Knowing how much muscle to remove and where to remove it from could reduce the likelihood of complications and suboptimal outcomes. In the present study, we employ an immersed boundary solver to model the effect of septal myectomy for ventricles with HOCM and demonstrate the potential of such an approach for surgical planning.

[1]Computational resources were provided by the National Institute of Computational Science under Tergrid grant number TG-CTS100002

8:13AM A13.00002 Investigation of Systolic Heart Murmurs with Computational Hemo-Acoustic Modeling , JUNG HEE SEO, RAJAT MITTAL, Johns Hopkins University, THEODORE ABRAHAM, Johns Hopkins Medicine — Detection and analysis of heart murmurs generated by abnormal blood flows can be used as a low cost, non-invasive routine screening for the heart disease. Phonocardiography is an approach which combines electronic sound detection with automated signal analysis for detecting abnormal heart murmurs but the current approach relies primarily on empirical statistical correlations and ignores the underlying physics of flow-induced sound generation and propagation. In the present study, we investigate the characteristics and generation mechanism of systolic heart murmurs associated with the obstructive hypertrophic cardiomyopathy (HOCM) using the computational fluid dynamics and acoustics modelings. The hemodynamic flow field in left ventricle outflow tract is simulated with the immersed boundary, incompressible Navier-Stokes solver, and the sound generated by the blood flow is modeled by the linearized perturbed compressible equations. The propagation of the sound through the surrounding tissues is also modeled by the linear structural wave equations. The simulated murmurs are analyzed for the timing, frequency, and intensity and the correlation with the hemodynamics is closely investigated to identify the source mechanisms.

8:26AM A13.00003 Analysis of intracardiac flows for Diastolic heart dysfunction[1] , RAJAT MITTAL, XUDONG ZHENG, VIJAY VEDULA, THEODORE ABRAHAM, Johns Hopkins University — Diastolic dysfunction (DD) is a common finding in a variety of cardiac diseases including hypertension, coronary disease and cardiomyopathy. Its prevalence increases with age and it manifests as incomplete or/and delayed ventricular relaxation and a compensatory stronger atrial contraction. DD is often associated with heart failure and contributes greatly to morbidity and hospitalizations especially in the elderly. In the current study, three-dimensional Navier-Stokes simulations are employed to investigate intracardiac flow behavior in normal and diseased hearts with DD. The endocardial surface of the left ventricle is represented by a generic simplified prolate-spheroid and the wall motion is driven by the ventricular volume change. Diastolic dysfunction in the heart is modeled by prescribing different E/A filling ratios. The dominant flow features, such as vortices and swirling structures and associated Eulerian and Lagrangian metrics are examined to gain insights into the flow physics of this disease.

[1]This research was supported by the National Science Foundation through TeraGrid Grant TG-CTS100002.

8:39AM A13.00004 Multiscale modeling and simulation of blood flow in coronary artery bypass graft surgeries[1] , SETHURAMAN SANKARAN, MAHDI ESMAILY MOGHADAM, ANDY KAHN, ALISON MARSDEN, UCSD — We present a computational framework for modeling and simulation of blood flow in patients who undergo coronary artery bypass graft (CABG) surgeries. We evaluate the influence of shape on the homeostatic state, cardiac output, and other quantities of interest. We present a case study on a patient with multiple CABG. We build a patient-specific model of the blood vessels comprised of the aorta, vessels branching from the top of the aorta (brachiocephalic artery and carotids) and the coronary arteries, in addition to bypass grafts. The rest of the circulatory system is modeled using lumped parameter 0D models comprised of resistances, compliances, inertances and elastance. An algorithm is presented that computes these parameters automatically given constraints on the flow. A Finite element framework is used to compute blood flow and pressure in the 3D model to which the 0D code is coupled at the model inlets and outlets. An adaptive closed loop BC is used to capture the coupling of the various outlets of the model with inlets, and is compared with a model with fixed inlet BC. We compare and contrast the pressure, flowrate, coronary perfusion, and PV curves obtained in the different cases. Further, we compare and contrast quantities of interest such as wall shear stress, wall shear stress gradients and oscillatory shear index for different surgical geometries and discuss implications of patient-specific optimization.

[1]I would like to acknowlege AHA for funding this work

8:52AM A13.00005 Fluid Structure Interaction Simulations of Pediatric Ventricular Assist Device Operation , CHRIS LONG, ALISON MARSDEN, YURI BAZILEVS, UCSD — Pediatric ventricular assist devices (PVADs) are used for mechanical circulatory support in children with failing hearts. They can be used to allow the heart to heal naturally or to extend the life of the patient until transplant. A PVAD has two chambers, blood and air, separated by a flexible membrane. The air chamber is pressurized, which drives the membrane and pumps the blood. The primary risk associated with these devices is stroke or embolism from thrombogenesis. Simulation of these devices is difficult due to a complex coupling of two fluid domains and a thin membrane, requiring fluid-structure interaction modeling. The goal of this work is to accurately simulate the hemodynamics of a PVAD. We perform FSI simulations using an Arbitrary Lagrangian-Eulerian (ALE) finite element framework to account for large motions of the membrane and the fluid domains. The air, blood, and membrane are meshed as distinct subdomains, and a method for non-matched discretizations at the fluid-structure interface is presented. The use of isogeometric analysis to model the membrane mechanics is also discussed, and the results of simulations are presented.

9:05AM A13.00006 Device specific analysis of neonatal aortic outflow cannula jet flows for improved cardiopulmonary bypass hemodynamics , PRAHLAD MENON, Carnegie Mellon University, FOTIS SOTIROPOULOS, University of Minnesota, AKIF UNDAR, Penn State Milton S. Hershey Medical Center, KEREM PEKKAN, Carnegie Mellon University — Hemodynamically efficient aortic outflow cannulae can provide high blood volume flow rates at low exit force during extracorporeal circulation in pediatric or neonatal cardiopulmonary bypass repairs. Furthermore, optimal hemolytic aortic insertion configurations can significantly reduce risk of post-surgical neurological complications and developmental defects in the young patient. The methodology and results presented in this study serve as a baseline for design of superior aortic outflow cannulae based on a novel paradigm of characterizing jet-flows at different flow regimes. *In-silico* evaluations of multiple cannula tips were used to delineate baseline hemodynamic performance of the popular pediatric cannula tips in an experimental cuboidal test-rig, using PIV. High resolution CFD jet-flow simulations performed for various cannula tips in the cuboidal test-rig as well as *in-vivo* insertion configurations have suggested the existence of optimal surgically relevant characteristics such as cannula outflow angle and insertion depth for improved hemodynamic performance during surgery. Improved cannula tips were designed with internal flow-control features for decreased blood damage and increased permissible flow rates.

9:18AM A13.00007 High Order Large Eddy Simulation (LES) of Flow in Idealized Total Cavopulmonary Connection (TCPC) , YANN DELORME, KAMESWARARAO ANUPINDI, DINESH SHETTY, ANNA-ELODIE KERLO, JUN CHEN, School of Mechanical Engineering, Purdue University, MARK RODEFELD, Indiana University School of Medicine, STEVEN FRANKEL, School of Mechanical Engineering, Purdue University — Irregular, transitional, and chaotic flow fields have been observed in previous experimental studies of the TCPC. Low-order numerical methods have been shown to be unable to capture these unsteady flow structures when compared to experimental flow studies. A high-order incompressible LES code is combined with a recent Immersed Boundary Method (IBM) to enable structured Cartesian grids to represent the TCPC geometry. Validation studies for canonical flows and recent PIV data for the TCPC case are shown. Analysis of the instantaneous vortical structure and mean statistics are presented to further elucidate the complex flow patterns and wall shear stress distributions. Recent studies proved that mechanical support through a Viscous Impeller Pump (VIP) can improve TCPC hemodynamics by increasing the pressure and the blood flow through the lungs. The inclusion of this novel VIP to power the Fontan circulation is also studied with further comparisons to PIV and HQ data. Finally, dynamic mode decomposition is used to study TCPC flow modes and particle transport is considered to assess thrombosis potential.

9:31AM A13.00008 Stereoscopic PIV of Powered Fontan Hemodynamics in Idealized Total Cavopulmonary Connection (TCPC)

ANNA-ELODIE KERLO, School of Mechanical Engineering, Purdue University, MARK RODEFELD, Indiana University School of Medicine, STEVEN FRANKEL, JUN CHEN, School of Mechanical Engineering, Purdue University — Stereoscopic Particle Image Velocimetry (SPIV) measurements are presented in an idealized TCPC geometry powered by a novel Viscous Impeller Pump (VIP) designed to provide mechanical assist to univentricular Fontan circulations. Instantaneous flow patterns and mean flow statistics are reported to provide insight into flow structure and wall shear stress. Dynamic mode decomposition is applied to SPIV data to extract relevant modes from the flow without and with the VIP present and operating. Previous experimental and computational studies showed excellent hemodynamic performance of the VIP: it stabilizes the four-way flow pattern without risk of obstruction to flow and augments flow in ideal pressure range (0-10 mmHg). This holds out the promises of providing temporary support for patients with failing Fontan circulations and ultimately reducing the 3-stage Fontan surgical procedures to one. Our SPIV data help to elaborate on VIP performance by analyzing detailed unsteady flow features and serve as a database for more advanced computational studies such as large eddy simulation.

9:44AM A13.00009 Elevated Shear Stress in Arteriovenous Fistulae: Is There Mechanical Homeostasis?[1]

PATRICK MCGAH, DANIEL LEOTTA, KIRK BEACH, ALBERTO ALISEDA, University of Washington — Arteriovenous fistulae are created surgically to provide access for dialysis in patients with renal failure. The current hypothesis is that the rapid remodeling occurring after the fistula creation is in part a process to restore the mechanical stresses to some preferred level (i.e. mechanical homeostasis). Given that nearly 50% of fistulae require an intervention after one year, understanding the altered hemodynamic stress is important in improving clinical outcomes. We perform numerical simulations of four patient-specific models of functioning fistulae reconstructed from 3D Doppler ultrasound scans. Our results show that the vessels are subjected to 'normal' shear stresses away from the anastomosis; about 1 Pa in the veins and about 2.5 Pa in the arteries. However, simulations show that part of the anastomoses are consistently subjected to very high shear stress (>10Pa) over the cardiac cycle. These elevated values shear stresses are caused by the transitional flows at the anastomoses including flow separation and quasiperiodic vortex shedding. This suggests that the remodeling process lowers shear stress in the fistula but that it is limited as evidenced by the elevated shear at the anastomoses. This constant insult on the arterialized venous wall may explain the process of late fistula failure in which the dialysis access become occluded after years of use.

[1]Supported by an R21 Grant from NIDDK (DK081823)

9:57AM A13.00010 Effects of Mixing on the Conductance Measurements of Artery Cross-Section Area

QING HAO, HYO WON CHOI, GHASSAN KASSAB, Indiana University Purdue University, Indianapolis — Turbulent dispersive mixing occurs during injection of saline into artery for the conductance catheter measurement of lumen cross section area (CSA). The objective of present study is to evaluate the accuracy of conductance catheter measurement in relation to dispersion. Computational fluid dynamics (CFD) simulations were performed to describe fluid flow, mass and electric field in artery with saline injected through different catheters and for different injection times. The simulation results showed that axial velocity profiles in all cases can reach stable after 0.5 sec and a recirculation zone exists distal to the saline injection site for Craya-Curtet number below critical value (0.7-0.98). A stable mixture ratio of saline and blood was reached in a distance of 8 artery diameters downstream of saline injection site. The results also showed that the accuracy of conductance catheter is affected by injection time, artery size, and conductance catheter location. The mixing ratio of saline and blood was shown to be a major factor affecting the accuracy of the conductance catheter measurement of CSA. A modification of electrical conductivity according to the mixing ratio improved the accuracy of conductance measurements and decreased the error to an acceptable level (<10%) regardless of the vessel size and injection flow rate.

Sunday, November 20, 2011 8:00AM - 10:10AM –

Session A14 Biofluids: Cellular I: Transport of Blood Cells 317

8:00AM A14.00001 Red blood cell damage by shear stress for different blood types[1]

GILAD ARWATZ, KATHERINE BEDKOWSKI, ALEXANDER SMITS — In surgical practice, blood damage caused by medical devices is often a limiting factor in the duration of an acute procedure or in chronic exposures such as hemodialysis. In order to establish guidelines for designing medical devices, a study was conducted to determine the relationship between shear stress and damage to red blood cells using a concentric Couette device. By measuring the hemolysis level for various shear stresses and exposure times, a non-dimensional relationship between shear stress and blood damage for different blood types was established.

[1]Funding provided by Princeton University's Project X.

8:13AM A14.00002 Transition to Non-Newtonian behavior of blood suspensions flowing in small tubes

BRUCE CASWELL, HUAN LEI, Brown University, DMITRY FEDOSOV, Forschungszentrum Julich, GEORGE KARNIADAKIS, Brown University — Blood flow in tubes is widely considered to be Newtonian down to diameters of about 200 microns. We have employed a multi-scale, Dissipative Particle Dynamics (DPD) model of the red blood cell (RBC) to investigate suspensions driven through small tubes (**diameters 20-150 microns).** The cross-stream stress gradient induces radial migration of the suspended RBCs resulting in the formation of a hematocrit (H) peak at the centerline, and at the wall a cell-free layer (CFL) whose edge is the point of maximum RBC distortion. This suggests that hard- sphere suspension theories will not capture well blood flow in tubes. For the larger tubes the velocity profiles beyond the CFL are essentially parabolic even though the core H is non-uniform. As the diameter decreases: (1) the CFL moves inward and the central H peak grows, but for the smallest (20 microns) the H peak is shifted off-center, (2) the bulk velocity profiles become similar to those of a shear-thinning non-Newtonian fluid. However, accurate modeling of the velocity field of the bulk flow in small tubes as a homogeneous non-Newtonian fluid can only be achieved if model parameters are taken to depend on tube diameter and pressure drop.

8:26AM A14.00003 Hydrodynamic forces on a wall-bound leukocyte due to interactions with flowing red cells

AMIR H. G. ISFAHANI, JONATHAN B. FREUND, University of Illinois at Urbana-Champaign — As part of both healthy and pathologically physiological mechanisms sphere-like white blood cells (leukocytes) adhere to the walls of small blood vessels. We use quantitative numerical simulations to compare the forces from flowing red blood cells on a wall-adhered leukocyte to a homogenized model of blood at the same flow conditions. We model the highly flexible red blood cells using a fast $O(N \log N)$ boundary integral formulation. These elastic membranes deform substantially but strongly resist surface dilatation. They enclose a higher than plasma viscosity hemoglobin solution. The no-slip condition is enforced on the stationary leukocyte as well as the vessel walls. Vessel diameters of 10 to 20 microns are studied. Different hematocrits, leukocyte shapes, and flow conditions are examined. In vessels comparable to the size of the cells, we show that the particulate character of blood significantly affects the magnitude of the forces that the leukocyte experiences, transiently increasing it well above the homogenized-blood prediction: for example, for a tube hematocrit of 25% and a spherical protrusion with a diameter 0.75 that of the tube, the average forces are increased by about 40% and the local forces by more than 100% relative to those expected for a blood model homogenized by its effective viscosity.

8:39AM A14.00004 Dynamics of Microcapsules and Red Blood Cells in Oscillating Shear Flow[1]

, MENGYE ZHAO, PROSENJIT BAGCHI, Rutgers University — In a recent experimental work, Dupire et al [PRL, **104**, 168101 (2010)] reported a nonperiodic behavior of the red blood cells suspended in a sinusoidally oscillating shear flow. The nonperiodic motion was characterized by intermittent cell swinging and tumbling. A theoretical model based upon the work of Keller and Skalak [JFM, **120**, 27 (1982)] for shape- preserving cells was shown to predict the nonperiodic motion that was highly sensitive to the initial conditions. In this talk, we present a three-dimensional numerical study of deformable capsules in sinusoidally oscillating shear flow in order to address if similar nonperiodic motion is observed when deformation is present. For initially oblate capsules, we observe two types of dynamics: a swinging motion in response to the altering flow direction that occurs at both high and low values of shear rate amplitudes, and a continuous/unidirectional tumbling motion that occurs at intermediate values. We obtain phase diagram that shows existence of two critical shear rates and two oscillation frequencies. Unlike Dupire et al, we do not find nonperiodic motion, but we find that the swinging/tumbling dynamics is sensitive to the initial condition.

[1]Supported by NSF.

8:52AM A14.00005 Modeling of Red Blood Cells and Related Spleen Function

, ZHANGLI PENG, Department of Materials Science and Engineering, Massachusetts Institute of Technology, IGOR PIVKIN, Institute of Computational Science, University of Lugano, MING DAO, Department of Materials Science and Engineering, Massachusetts Institute of Technology — A key function of the spleen is to clear red blood cells (RBCs) with abnormal mechanical properties from the circulation. These abnormal mechanical properties may be due to RBC aging or RBC diseases, e.g., malaria and sickle cell anemia. Specifically, 10% of RBCs passing through the spleen are forced to squeeze into the narrow slits between the endothelial cells, and stiffer cells which get stuck are killed and digested by macrophages. To investigate this important physiological process, we employ three different approaches to study RBCs passage through these small slits, including analytical theory, Dissipative Particle Dynamics (DPD) simulation and Multiscale Finite Element Method (MS-FEM). By applying the analytical theory, we estimate the critical limiting geometries RBCs can pass. By using the DPD method, we study the full fluid-structure interaction problem, and compute RBC deformation under different pressure gradients. By employing the MS-FEM approach, we model the lipid bilayer and the cytoskeleton as two distinct layers, and focus on the cytoskeleton deformation and the bilayer-skeleton interaction force at the molecular level. Finally the results of these three approaches are compared to each other and correlated to the experimental observations.

9:05AM A14.00006 Phase Diagram and Breathing Dynamics of Red Blood Cell Motion in Shear Flow[1]

, PROSENJIT BAGCHI, ALIREZA YAZDANI, Rutgers University — We present phase diagrams of red blood cell dynamics in shear flow using three-dimensional numerical simulations. By considering a wide range of shear rate and interior-to-exterior fluid viscosity ratio, it is shown that the cell dynamics is often more complex than the well-known tank-treading, tumbling and swinging motion, and is characterized by an extreme variation of the cell shape. We identify such complex shape dynamics as 'breathing' dynamics. During the breathing motion, the cell either completely aligns with the flow direction and the membrane folds inward forming two cusps, or, it undergoes large swinging motion while deep, crater-like dimples periodically emerge and disappear. At lower bending rigidity, the breathing motion occurs over a wider range of shear rates, and is often characterized by the emergence of a quad-concave shape. The effect of the breathing dynamics on the tank-treading-to-tumbling transition is illustrated by detailed phase diagrams which appear to be more complex and richer than those of vesicles. In a remarkable departure from classical theory of nondeformable cells, we find that there exists a critical viscosity ratio below which the transition is dependent on shear rate only.

[1]Supported by NSF

9:18AM A14.00007 Red blood cell clusters in Poiseuille flow

, GIOVANNI GHIGLIOTTI, University of British Columbia, Vancouver, Canada, HASSIB SELMI, Ecole Polytechnique de Tunisie - La Marsa, Tunisia, CHAOUQI MISBAH, Universite de Grenoble-CNRS - Grenoble, France, LASSAAD ELASMI, Ecole Polytechnique de Tunisie - La Marsa, Tunisia — We present 2D numerical simulations of sets of vesicles (closed bags of a lipid bilayer membrane) in a parabolic flow, a setup that mimics red blood cells (RBCs) in the microvasculature. Vesicles, submitted to sole hydrodynamical interactions, are found to form aggregates (clusters) of finite size. The existence of a maximal cluster size is pointed out and characterized as a function of the flow intensity and the swelling ratio of the vesicles. Moreover bigger clusters move at lower velocity, a fact that may prove of physiological interest. These results quantify previous observations of the inhomogeneous distribution of RBCs in vivo (Gaehtgens et al., Blood Cells 6 - 1980). An interpretation of the phenomenon is put forward based on the presence of boli (vortices) between vesicles. Both the results and the explanation can be transposed to the three-dimensional case.

9:31AM A14.00008 Microfluidic investigation of the effects of oxidative stress on mechanotransduction in red blood cells

, N.F. ZENG, W.D. RISTENPART, University of California at Davis — Recent work has suggested that RBCs are able to sense and respond to small changes in their environment through post translational modifications (PTMs) in membrane proteins. Because oxidative stress is an important driving force to induce PTMs, the effects of oxidative stress on membrane deformability, lipid peroxidation, and cytoskeletal/hemoglobin crosslinking have been studied extensively. However, experimental work to date on the effects of oxidative stress on RBC mechanotransduction has been limited to applied forces dissimilar to those experienced by RBCs in vivo. Here we investigate the dynamics of shear-induced mechanotransduction in RBCs subjected to varying degrees of oxidative stress by using hydrogen peroxide as a generator of oxidizing radicals. We use a microfluidic platform to impose precisely defined fluid flows that mimic in vivo conditions. The RBCs are visualized passing through a narrow constriction using high speed video at 15,000 frames per second, and quantitative hematological information including cell elongation, rotation and velocity are extracted via custom image analysis algorithms. We demonstrate that oxidative stress significantly alters the dynamic behavior of the RBCs under flow conditions, and we discuss the implications for the consequent effects on mechanotransductive vasodilatory signaling.

9:44AM A14.00009 Stability of red cells flowing in a narrow tube[1]

, NATALIE BEAMS, JONATHAN FREUND, University of Illinois at Urbana-Champaign — Red blood cells are well known to line up in an orderly arrangement when forced to flow through a narrow capillary-scale round tube (diameter $\leq 8\mu$m). However, in slightly larger tubes this order can break down, resulting in apparently chaotic flow. We investigate this breakdown using a high-fidelity boundary integral solver for flowing blood cells. This solver has been validated for both the flow of organized and highly deformed cells in narrow tubes and for more random flow in larger tubes. Our studies focus on a family of cases with 8 red cells, each discretized with spherical harmonics. The cells are modeled as elastic shells enclosing a viscous fluid. Studying the development of instabilities using ad hoc perturbation techniques as well as non-normal modal analysis, we show a strong increase in instability for larger tube diameters. Increasing the cell interior viscosity is also observed to increase the amplification of perturbations.

[1]Supported by NSF

9:57AM A14.00010 2-D Model for Normal and Sickle Cell Blood Microcirculation , YONATAN TEKLEAB, WESLEY HARRIS, Massachusetts Institute of Technology — Sickle cell disease (SCD) is a genetic disorder that alters the red blood cell (RBC) structure and function such that hemoglobin (Hb) cannot effectively bind and release oxygen. Previous computational models have been designed to study the microcirculation for insight into blood disorders such as SCD. Our novel 2-D computational model represents a fast, time efficient method developed to analyze flow dynamics, O_2 diffusion, and cell deformation in the microcirculation. The model uses a finite difference, Crank-Nicholson scheme to compute the flow and O_2 concentration, and the level set computational method to advect the RBC membrane on a staggered grid. Several sets of initial and boundary conditions were tested. Simulation data indicate a few parameters to be significant in the perturbation of the blood flow and O_2 concentration profiles. Specifically, the Hill coefficient, arterial O_2 partial pressure, O_2 partial pressure at 50% Hb saturation, and cell membrane stiffness are significant factors. Results were found to be consistent with those of Le Floch [2010] and Secomb [2006].

Sunday, November 20, 2011 8:00AM - 9:57AM –
Session A15 Rayleigh-Benard Convection I: Instability and Oscillations 318

8:00AM A15.00001 Onset of Three-Dimensional Thermal Convection in a Rectangular Cavity , MITSUAKI FUNAKOSHI, YOSHINARI FUKAZAWA, Graduate School of Informatics, Kyoto University — The onset of three-dimensional thermal convection of a fluid in a rectangular cavity is numerically studied, under the assumption that its all walls are rigid and of perfect thermal conductance with vertically linear temperature profile. The dependence of the critical Rayleigh number and the symmetry of the most unstable mode on the aspect ratios of the cavity is obtained.

8:13AM A15.00002 Numerical Investigations of Convection[1] , BRANDON CLOUTIER, PAUL RIGGE, JARED WHITEHEAD, BENSON MUITE, University of Michigan, HANS JOHNSTON, University of Massachusetts Amherst — We report on high resolution numerical studies of infinite Prandtl number convection using a simplified model with relevance to the motion of the Earth's mantle. Our model uses Navier-Stokes equations with the Boussinesq approximation and free slip velocity boundary conditions that is driven soley by internal heating. The 2D simulations are calculated using pseudospectral Fourier and Chebyshev methods. We examine the transition from conduction to steady convection, to unsteady laminar convection and lastly to chaotic convection. As the heating rate is increased, we report on the relationship between the non- dimensional heat Rayleigh number (proportional to the heating rate) and the averaged temperature (spatially and temporally). We also compare different aspect ratios (width to height) to see the impact this change has on our system.

[1]Some of this work was performed on Teragrid resources supported by award TG-CTS110010.

8:26AM A15.00003 High Rayleigh number porous convection , DUNCAN HEWITT, JOHN LISTER, JEROME NEUFELD, DAMTP, University of Cambridge — Convective flow in porous media undergoes a transition around Rayleigh number $Ra = 1300$, from predominantly large-scale (quasi-) periodic rolls to vigorous columnar exchange flow driven by unsteady plume formation in boundary layers. The dynamics of these structures determine the flux of heat or solute through the system, as described by the Nusselt number Nu. This is of particular interest for understanding how convection affects the rate of dissolution of sequestered CO_2 in a saline aquifer. High resolution 2D numerical simulations of porous media Rayleigh-Benard convection are presented, which show that, for $2000 < Ra \leq 40000$, the dependence of Nu on Ra is slightly weaker than the classical scaling $Nu \propto Ra$. However, the numerical results strongly suggest that the classical scaling is attained asymptotically, contrary to previous indications. The relationship between $Nu(Ra)$ and the global structure of the flow is discussed. This structure is characterised in the interior by vertical columnar exchange flow across the height of the box at a (statistically) regular wavelength, and near the boundaries by short wavelength boundary layer instabilities driving proto-plume growth and entrainment into the interior mega-plume flow. The dynamics of the system at all scales are analysed, with the aim of understanding the dominant physical processes.

8:39AM A15.00004 Onset of Convection in a Diffusive Miscible Displacement Porous Media Flow with Viscosity Contrast , DON DANIEL, AMIR RIAZ, University of Maryland, College Park — One of the typical yet important type of flows in porous media is the case in which two miscible fluids begin to interact with each other due to the gravitational instability that exists due to the density differences between the underlying fluids. This problem has been extensively studied in literature. However many of these studies, have not considered the role the viscosity difference can play during the onset of convection. The onset of convection cannot be captured by most typical linear stability analysis, due to the continuous spectrum of the concerned operator. We convert our governing equations such that they effectively involve a discrete spectrum which in turn provides more accurate and reliable platform for conducting linear stability analysis. In order to verify our analysis, we compare the obtained growth rates with those obtained by fully resolved non-linear 2D simulations.

8:52AM A15.00005 Buoyancy driven flows in porous media: Effect of chemical reactions and transverse velocities , S. HOSSEIN HEJAZI, JALEL AZAIEZ, University of Calgary — Density or viscosity mismatch among different solutions in porous media may result in interfacial instability. Many underground reservoir flow processes such as in-situ upgrading of heavy oil, in-situ water remediation and geological storage of carbon dioxide involve chemical reactions and natural transverse flows. As a result, further complicated hydrodynamical instabilities may be encountered. In this study, a $2D$ vertical porous medium saturated with a reactant A placed at the top of another reactant B is considered where a chemical product C is formed. All chemical components are assumed to have different physical properties namely the density and the viscosity. Moreover, a transverse flow is introduced parallel to the initial reactant interface. In the limit of small times the stability characteristics of the flow are examined. Thereafter, the evolutions of unstable modes are analyzed by conducting full nonlinear simulations. Chemical production concentration iso-surfaces are tracked and analyzed in time. A quantitative analysis is performed in terms of the total amount of chemical product. A physical discussion on how fluid reactivity and presence of transverse flows can affect the fate of hydrodynamical instabilities is presented.

9:05AM A15.00006 Experiments and High-resolution Simulations of Density and Viscosity Feedbacks on Convective Mixing[1] , JUAN J. HIDALGO, MIT, Cambridge, MA, USA, JAIME FE, University of A Coruña, A Coruña, Spain, CHRISTOPHER W. MACMINN, LUIS CUETO-FELGUEROSO, RUBEN JUANES, MIT, Cambridge, MA, USA — Dissolution by convective mixing is one of the main trapping mechanisms during CO_2 sequestration in saline aquifers. Initially, the buoyant CO_2 dissolves into the underlying brine by diffusion. The CO_2-brine mixture is denser than the two initial fluids, leading to a Rayleigh-Bénard-type instability known as convective mixing, which greatly accelerates CO_2 dissolution. Although this is a well-known process, it remains unclear how convective mixing scales with the governing parameters of the system and its impact on the actual mixing of CO_2 and brine. We explore the dependence of the CO_2 dissolution flux on the nonlinearity of the density and viscosity of the fluid mixture by means of high-resolution numerical simulations and laboratory experiments with an analogue fluid system (water and propylene glycol). We find that the value of the concentration for which the density of the mixture is maximum, and the viscosity contrast between the fluids, both exert a powerful control on the convective flux. From the experimental and simulation results, we obtain the scaling behavior of convective mixing, and clarify the role of nonlinear density and viscosity feedbacks.

[1]JJH acknowledges the support from the FP7 Marie Curie Actions of the European Commission, via the CO2-MATE project (PIOF-GA-2009-253678).

9:18AM A15.00007 The intermediate regime of convection across a permeable membrane , VIJAYA RAMA REDDY GUDLA, BHEL Corporate R&D, Hyderabad, India, B.A. PUTHENVEETTIL, Dept. of Applied Mechanics, IIT Madras, Chennai, India — In an arrangement of brine over water across a horizontal permeable membrane, where the unstable density difference across the membrane initiates convection, we discover a new convection regime where the Sherwood number scales approximately as the Rayleigh number. Inferring from the planforms of plume pattern and the estimates of velocity through the membrane, we show that such a regime occurs when advection balances diffusion in the membrane pore. Utilizing mass balance and symmetry assumptions in the top and bottom fluids, convection-diffusion equation for the membrane pore is solved to obtain concentration drops across and in the boundary layers above and below the membrane. With the observation that the normalized net flux is constant in the new regime, an expression for the flux scaling in the new regime is derived. The scaling matches with the experiments and has correct asymptotes in the advection and diffusion regimes (Puthenveettil & Arakeri, JFM, V542, 2005; V609, 2008). The plume spacings in the new regime are distributed lognormally, and their mean follow the trend in the advection regime.

9:31AM A15.00008 Rayleigh-Benard instability in multicomponent mixtures with the Soret effect[1] , ILYA RYZHKOV, Institute of Computational Modelling SB RAS — Convection in multicomponent mixtures can show a variety of flow patterns due to several heat and mass transfer mechanisms: convection, heat conduction, main and cross diffusion, and the Soret effect. Convective stability of multicomponent fluids has not been widely investigated so far. The use of simplifying assumptions (e.g. the absence of cross-diffusion) may lead to the disagreement between theory and experiment. We study the stability of a plane multicomponent fluid layer heated from above/below in gravity field. In the basic state, the fluid is at rest and temperature gradient induces concentration gradients due to the Soret effect. The problem is reduced to that without cross-diffusion and Soret effect by a special transformation. Several types of boundary conditions are considered: 1) free, permeable 2) rigid, permeable 3) rigid, impermeable. The theorems, which generalize the exchange of stability principle to multicomponent fluids, are proved for boundary conditions 1 and 2. An explicit formula for critical Rayleigh numbers is obtained for boundary conditions 1. The stability problem for boundary conditions 3 was solved numerically for a ternary mixture. The stability maps are constructed in a wide range of parameters.

[1]The work is supported by the Interdisciplinary project of SB RAS N 116 and Krasnoyarsk Science Foundation.

9:44AM A15.00009 Convective filtration near solid inclusion in a fluid heated from above[1] , TATYANA LYUBIMOVA, Institute of Continuous Media Mechanics UB RAS, Perm, Russia, DMITRIY LYUBIMOV, Perm State University, Perm, Russia — Convective filtration near a highly conductive cylindrical solid inclusion in porous medium saturated by a fluid is studied for the case of heating from above. Convective flow is absent far from inclusion and near inclusion conditions for the conductive state are not satisfied and convection arises. The study is performed in the framework of Darcy-Boussinesq approximation. Two-dimensional flow uniform along the cylinder axis is considered. We use Oseen-like approach where, however, quasi-linearization is applied to the nonlinear terms in the energy equation is not in the momentum. In the first approximation we obtain a system of linear equations with the Rayleigh number as a parameter. An asymptotic representation of solutions uniformly valid at all distances from the inclusion is obtained for small values of Rayleigh number. It is shown that convective flow at the distances from the inclusion larger than the inclusion size has the form of horizontal jets directed away from the body and slowly expanding with the increase of distance from the body. The compensational flow is not of the jet type. The case of finite thermal conductivity of the inclusion is studied in the framework of similar approach.

[1]The work is supported by RFBR (grant 10-01-00679).

Sunday, November 20, 2011 8:00AM - 10:10AM –
Session A16 Experimental Techniques I 319

8:00AM A16.00001 An experimental investigation of the influence of particle displacement and image number on PIV uncertainty for two turbulent flows , RALPH BUDWIG, University of Idaho, CHRISTOPHER HOCUT, University of Notre Dame — In order to test sensitivity to particle displacement, the mean velocity and turbulent fluctuations were calculated from six sets of stereoscopic PIV images, each with a different average particle displacements. The first set of images had an average particle displacement of five pixels and in each subsequent set the displacement was increased by five pixels. In order to test sensitivity to image number, the number of images used for ensemble averaging was varied from 1000 to 4000 The first flow tested was a nearly fully developed turbulent open channel flow (moderate turbulence levels). The second was the turbulent wake immediately behind a circular cylinder that extended from just above the channel bottom through the water free surface (high turbulence levels). Comparisons to ADV measurements were made at selected points in both flows. [1]Center for Ecohydraulics Research [2]Environmetnal Fluid Dynamics Laboratory

8:13AM A16.00002 ABSTRACT WITHDRAWN —

8:26AM A16.00003 A postprocessing method for finding flow derivatives on an unstructured grid using PTV data , MICAH PAUL, DANA DABIRI, University of Washington — Existing methods of processing particle tracking velocimetry (PTV) data to determine derivative information, such as shear and vorticity, require that the scattered velocity data be interpolated onto a uniform grid. While this makes calculations of flow derivatives simple, it necessarily introduces error. Here, a process is proposed which allows for the calculation of differential properties on unstructured grids. The proposed method first uses an anisotropic diffusion smoothing technique to denoise the velocity data, and then uses a natural neighbors-based Laplace interpolation to calculate local velocity derivatives. Results using this technique are discussed.

8:39AM A16.00004 3 dimension 3 coordinate velocity measurements of electrothermal vortex using Interference micro Particle Tracking Velocimetry , CRAIG SNOEYINK, STEVE WERELEY, Purdue University — We present velocity measurements of an electrothermal vortex using Interference micro Particle Tracking Velocimetry (I-μPTV). These measurements are compared to results obtained using Astigmatism micro-Particle Tracking Velocimetry (A-μPTV). We will also demonstrate several of the advantages of this system such as an increased measurement volume thickness as well as the ability to use small particles (< 1 μm diameter).

8:52AM A16.00005 Color-Coded Three-Dimensional Particle Tracking Velocimetry for Micro-Flow Applications[1] , WEI-HSIN TIEN, DANA DABIRI, University of Washington, JAY HOVE, University of Cincinnati — A color-coded three-dimensional particle image velocimetry is successfully adapted into a microscope setup using a 10X magnification lens. 3 high-power LED with different wavelengths are used as light sources for each color-coded pinhole, and a color separation algorithm based on decorrelation stretch is developed to account for the multiple exposure problem caused by the color filters. Overlapped particle images are then identified with a cascade correlation method, and the triplets are matched to reconstruct the 3D particle locations with a calibration-based epi-polar line search method. The velocity field is reconstructed by a vision-based particle tracking algorithm. With the ability to tack particles in higher particle densities, the experimental setup is used to image an 800 x 800 x 500 micrometer volume of an accelerating backward-facing step micro-channel flow.

[1]This work is supported by the National Institute of Health (NIH)

9:05AM A16.00006 Split view Time-resolved PIV with a CW laser for 3-D measurements of planar velocity field[1] , AMIR ELZAWAWY, Graduate Center of CUNY, YIANNIS ANDREOPOULOS, City College of New York — The demand to increase the temporal resolution of Stereo-PIV systems used in the measurement of highly unsteady flow fields is limited by the low repetition rate of the pulsed lasers and cameras. The availability of high-frame-rate digital cameras and CW lasers opens new possibilities in the development of continuous PIV systems with increased temporal resolution. The present setup consists of a single high-frame-rate camera which can accommodate two simultaneous stereo view images of the deforming fluid on its CMOS sensor obtained by using four different planar mirrors, appropriately positioned This approach offers several advantages over traditional systems with two different cameras. First, it provides identical system parameters for the two views which minimize their differences and thus facilitating robust stereo matching. Second, it eliminates any need of synchronization between both cameras and the laser. And third its cost is substantially lower than the cost of a system with two cameras. The development of the technique will be described and the results of qualification tests in several wind tunnel flows will be presented and discussed.

[1]Sponsored by NSF Grant #1033117.

9:18AM A16.00007 Accuracy of standard image processing and PIV analysis applied to x-ray velocimetry , ELIZABETH VOIGT, RODERICK LA FOY, KAMEL FEZZAA, WAH-KEAT LEE, PAVLOS VLACHOS, Virginia Tech — Particle image velocimetry (PIV) using x-ray illumination is a recently developed technique providing a new capability to study low-Reynolds number opaque flows such as blood flow in microvessels. Preliminary work in x-ray PIV by several groups has used image processing and PIV techniques originally developed for optical PIV images, although significant differences exist between standard PIV images and x-ray PIV images. This work quantitatively assesses the validity of standard image processing and PIV techniques for x-ray images. Data were acquired for steady Poiseuille flow using hollow glass microspheres as tracers. Images were pre-processed using techniques including mean subtraction, Fourier filter, phase retrieval, and inversion filter, and velocity fields were computed using standard cross-correlation and robust phase correlation. A metric of image quality was found that reliably predicts the best of the pre-processed image sets. Quantitative comparison of velocity profiles with the Poiseuille solution indicated that robust phase correlation outperforms standard cross correlation for x-ray images.

9:31AM A16.00008 Developing an ultrasound correlation velocimetry system[1] , GERRIT SURUP, CHRISTOPHER WHITE, University of New Hampshire, UNH TEAM — The process of building an ultrasound correlation velocimetry (UCV) system by integrating a commercial medical ultrasound with a PC running commercial PIV software is described and preliminary validation measurements in pipe flow using UCV and optical particle image velocimetry (PIV) are reported. In principles of operation, UCV is similar to the technique of PIV, differing only in the image acquisition process. The benefits of UCV are that it does not require optical access to the flow field and can be used for measuring flows of opaque fluids. While the limitations of UVC are the inherently low frame rates (limited by the imaging capabilities of the commercial ultrasound system) and low spatial resolution, which limits the range of velocities and transient flow behavior that can be measured.

[1]The support of the NSF (CBET0846359, grant monitor Horst Henning Winter) is gratefully acknowledged.

9:44AM A16.00009 Power Spectrum Estimation of Randomly Sampled LDA Signals , PREBEN BUCHHAVE, CLARA VELTE, Technical University of Denmark, WILLIAM GEORGE, Imperial College — Power spectra from LDA data must be computed using the theory of burst-mode LDA processing in which the sampling bias is corrected by means of the measured residence time [1,2]. However, the validity of this method has been disputed, and alternative methods have been proposed, partly on theoretical grounds, partly with reference to practical problems with the measurement of the residence time [3]. We present power spectra computed by a number of different algorithms (i.a. the block averaged direct spectral estimator, the Fourier transform of the slotted autocovariance estimate and the sample-and-hold method) using simulated data from 3-D turbulent velocity, and we show that only residence time weighted algorithms give the correct answer in all situations. We also demonstrate clearly effect of using the wrong algorithms by processing some single frequency spectra. We confirm these conclusions by evaluating power spectra from a turbulent jet [4]. The power spectra from the jet data show good agreement with the results from simulated data. [1] Buchhave, Ph.D dissertation 1979 [2] Buchhave, George and Lumley, Ann. Rev. Fluid Mech. 1979 [3] Albrecht, Borys, Damaschke and Tropea, Springer Verlag pp. 552, 2003 [4] Velte, Ph.D. dissertation 2010

9:57AM A16.00010 An Experimental Investigation on the Wake Interference of Multiple Wind Turbines over Complex Terrains , HUI HU, ZIFENG YANG, AHMET OZBAY, PARTHA SARKAR, Iowa State University — We report an experimental study to investigate the wake interferences of multiple wind turbines sited over complex terrains in order to elucidate underlying physics to explore/optimize design paradigms of wind turbines sited over complex terrains for higher power yield and better durability. The experiments were conducted in a wind tunnel with an array of wind turbine models sited over a flat (baseline case) and complex terrains with non-homogenous surface winds. In addition to measuring dynamic wind loads (both forces and moments) and the power outputs of the wind turbine models, advanced flow diagnostic techniques such as digital Particle Image Velocimetry (PIV) will be used to conduct detailed flow field measurements to quantify the flow characteristics of the surface winds and wake interferences among multiple wind turbines over flat (baseline case) and complex terrains. The detailed flow field measurements were correlated with the wind load measurements to elucidate the underlying physics associated with turbine power generation and fatigue loads acting on wind turbines.

Sunday, November 20, 2011 8:00AM - 10:10AM –
Session A17 Surface Tension I 320

8:00AM A17.00001 Numerical simulation of drop dynamics with soluble surfactant , MICHAEL BOOTY, MICHAEL SIEGEL, New Jersey Institute of Technology, QIMING WANG, University of British Columbia — We use a hybrid numerical method that combines the boundary integral method with an asymptotic reduction of surfactant solubility effects neighboring a drop interface in the large bulk Peclet number limit. This provides an accurate and efficient means of simulating drop dynamics with soluble surfactant. The influence of surfactant solubility on steady-state solution branches for a drop that is stretched in an axisymmetric extensional flow is presented together with the influence of various parameters, including the capillary and Biot numbers, on unsteady drop evolution and tipstreaming.

8:13AM A17.00002 Dispersion of Particles on Fluid-Liquid Interfaces , B. DALAL, S. GURUPATHAM, M. HOSSAIN, I. FISCHER, P. SINGH, New Jersey Institute of Technology, D. JOSEPH, University of Minnesota — This talk is concerned with the dispersion of particles on the fluid-liquid interface. In our previous studies we have shown that when small particles, e.g., flour, pollen, etc., come in contact with an air-liquid interface, they disperse in a manner that appears explosive. This is due to the fact that the capillary force pulls particles into the interface causing them to accelerate to a relatively-large velocity. The motion of particles in the direction normal to the interface is inertia dominated, and so they oscillate vertically about the equilibrium position before coming to a stop under viscous drag. This causes a radially-outward lateral flow on the interface that causes nearby particles to move away. In experiments the strength of the lateral flow was measured using tracer particles that were placed on the interface for this purpose. The dispersion on a liquid-liquid interface was relatively weaker than on an air-liquid interface, and occurred over a longer period of time. This partly was a consequence of the fact that particles became separated while sedimenting through the upper liquid and reached the interface over a time interval that lasted for several seconds. The rate of dispersion depended on the size of particles, the particle and liquids densities, the viscosities of the liquids involved, and the contact angle.

8:26AM A17.00003 Drops on elastic tracks , CAMILLE DUPRAT, Princeton University, SUZIE PROTIERE, CNRS/IJLRA-Universite Paris 6, ALEXANDER BEEBE, HOWARD STONE, Princeton University — Fibrous media are ubiquitous functional materials, which often consist of flexible high aspect ratio fibers that can easily deform under capillary forces with many industrial and ecological consequences. We study the influence of a mist of droplets on an elastic array of fibers by considering a finite volume drop on a pair of two flexible fibers, clamped at one end and free to deflect at the other. The elastocapillary deformation of the fibers leads to the spontaneous motion of the drop toward the free ends. The drop either remains compact with minimal spreading or spreads into a long liquid column that coalesces the fibers. We find that there is a critical volume of liquid, hence a critical drop size, above which this coalescence does not occur, and we identify another drop size which maximizes spreading, thus liquid capture. Experimental results and mathematical models will be presented and compared. These ideas are applicable to a wide range of fibrous materials, as we illustrate with quantitative examples for feathers, beetle tarsi, sprays and microfabricated systems.

8:39AM A17.00004 Streaming flows and propulsion by vertically oscillating free surfaces[1] , JUNQI YUAN, SUNG KWON CHO, University of Pittsburgh, UNIVERSITY OF PITTSBURGH TEAM — Vertical oscillations of the free surface generated by AC electrowetting can be used to propel mini-scale floating objects. In this presentation, detailed mechanisms in propulsion are discussed. Flow visualization shows that the vertical oscillations of free surface generate a quasi-steady streaming flow. The overall flow pattern is that the fluid is tangentially drawn from the sides of the electrode and is ejected normal to the electrode. It seems that this streaming flow is responsible for pushing the electrode in an opposite direction to the normal flow. Additionally, the effects of amplitude and frequency of vertical oscillation on the streaming flow pattern and propulsion efficiency are also presented. Finally, it is discussed that the depth of free surface is another key parameter that affects propulsion strength.

[1]This work is in part supported by the National Science Foundation

8:52AM A17.00005 Fabrication of Superhydrophobic Fiber Coatings by DC-Biased AC-Electrospinning[1] , M. GAD-EL-HAK, F.O. OCHANDA, M.A. SAMAHA, H. VAHEDI TAFRESHI, G.C. TEPPER, Virginia Commonwealth University — Mesh-like fiber mats of polystyrene (PS) were deposited using DC-biased AC-electrospinning. Superhydrophobic surfaces with water contact angles greater than 150° and gas fraction values of up to 97% were obtained. A Rheological study was conducted on these fiber surfaces and showed a decrease in shear stress when compared with a noncoated surface (no slip), making them excellent candidates for applications requiring the reduction of skin-friction drag in submerged surfaces. We have also shown that addition of a second, low-surface energy polymer to a solution of PS can be used to control the fiber internal porosity depending on the concentration of the second polymer. Contact-angle measurements on mats consisting of porous and nonporous fibers have been used to evaluate the role of the larger spaces between the fibers and the pores on individual fibers on superhydrophobicity.

[1]Financial support from the Defense Advanced Research Projects Agency (DARPA), contract number W91CRB-10-1-0003, is acknowledged.

9:05AM A17.00006 Drop Impingement on Highly Wetting Micro/Nano Porous Surfaces , CULLEN BUIE, YOUNGSOO JOUNG, Massachusetts Institute of Technology — Recently, we developed a novel fabrication method using a combination of electrophoretic deposition (EPD) and break down anodization (BDA) to achieve highly wetting nanoporous surfaces with microscale features. In this study we investigate droplet impingement behavior on these surfaces as a function of impact velocity, droplet size, and liquid properties. We observe impingement modes we denote as "necking" (droplet breaks before full penetration in the porous surface), "spreading" (continuous wicking into the porous surface), and "jetting" (jets of liquid emanate from the edges of the wicking liquid). To predict the droplet impingement modes, we've developed a non-dimensional parameter that is a function of droplet velocity, dynamic viscosity, effective pore radius and contact angle. The novel dimensionless parameter successfully predicts drop impingement modes across multiple fluids. Results of this study will inform the design of spray impingement cooling systems for electronics applications where the "spreading" mode is preferred.

9:18AM A17.00007 Armored droplets from particle rafts and an application to environmental remediation , HOWARD A. STONE, Princeton University, MANOUK ABKARIAN, Universite Montpellier, SUZIE PROTIERE, Institut Jean le Rond d'Alembert, Paris, JEFF ARISTOFF, Princeton University — Applications such as handling hazardous materials or containing chemical reactions involve encapsulating one fluid phase by a shell. In addition to surfactants, a variety of physicochemical approaches have been studied where nano- and/or micron-size particles organize at fluid-fluid interfaces to form some sort of shell. Gravity typically plays an insignificant role compared to surface forces in establishing such "armored interfaces." More generally, however, it is well known that capillary and gravitational forces cause particles trapped at interfaces to self-assemble and organize into raft-like structures. We describe gravity-driven instabilities of particle rafts and show how the "interfacial granular dynamics" lead to encapsulation strategies involving stable particle-armored droplets. Experimental results are compared with mathematical models of the composite objects in order to establish a quantitative description of our observations. The application of these ideas to environmental remediation will be described.

9:31AM A17.00008 Capillary attraction between floating cylinders , HARISH N. DIXIT, GEORGE M. (BUD) HOMSY, Mathematics Department, University of British Columbia — Capillary attraction between floating particles is a phenomenon of everyday experience and causes the particles at fluid interfaces to agglomerate. This is sometimes called the "Cheerios effect." The calculation of the force of attraction for even simple particle shapes is enormously difficult due to the presence of nonlinearity in the equations governing the interface shape. Most of the earlier approaches to obtained analytical expression for the force of attraction rely on the "superposition principle" by linearizing the Young-Laplace equation. We use a systematic perturbation technique in the limit of small Bond number. We present asymptotic solutions in a static configuration obtained for two problems: (i) attraction between two isolated cylinders, and (ii) attraction for an infinite array of cylinders. It is also found that the background curvature of an interface modifies the force of attraction. These forces are important in many technological applications: one such example is the problem of dip-coating of plates which will be discussed during the meeting.

9:44AM A17.00009 Coalescence-induced droplet actuation[1] , MATHIEU SELLIER, University of Canterbury, CLAUDE VERDIER, CNRS, VOLKER NOCK, University of Canterbury — This work investigates a little explored driving mechanism to actuate droplets: the surface tension gradient which arises during the coalescence of two droplets of liquid having different compositions and therefore surface tensions. The resulting surface tension gradient gives rise to a Marangoni flow which, if sufficiently large, can displace the droplet. In order to understand, the flow dynamics arising during the coalescence of droplets of different fluids, a model has been developed in the lubrication framework. The numerical results confirm the existence of a self-propulsion window which depends on two dimensionless groups representing competing effects during the coalescence: the surface tension contrast between the droplets which promotes actuation and species diffusion which tends to make the mixture uniform thereby anihilating Marangoni flow and droplet motion. In parallel, experiments have been conducted to confirm this self-propulsion behaviour. The experiment consists in depositing a droplet of distilled water on a "hydrophilic highway." This stripe was obtained by plasma-treating a piece of PDMS shielded in some parts by glass coverslips. This surface functionalization was found to be the most convenient way to control the coalescence. When a droplet of ethanol is deposited near the "water slug," coalescence occurs and a rapid motion of the resulting mixture is observed.

[1]The support of the Dumont d'Urville NZ-France Science & Technology program is gratefully acknowledged.

9:57AM A17.00010 Immiscible surfactant droplets on thin liquid films: Spreading dynamics, subphase expulsion, oscillatory instabilities and the effect of spatial confinement , DAVID SINZ, MYROSLAVA HANYAK, ANTON DARHUBER, Mesoscopic Transport Phenomena Group, Department of Applied Physics, Eindhoven University of Technology — After deposition of immiscible, surface-active liquids on thin liquid films of higher surface tension, Marangoni stresses thin the liquid film around the surfactant droplet and induce a flow directed away from the surfactant source. We present a combined experimental and numerical study elucidating a variety of aspects of this phenomenon. In a radially symmetric configuration we investigated an oscillatory instability, caused by temporary trapping and subsequent release of subphase liquid from underneath the surfactant droplet. We provide evidence that this expulsion has a pronounced effect on the spreading dynamics. Using chemical surface patterning we study the effect of spatial confinement of the subphase liquid on the spreading dynamics. The lateral confinement induces non-uniform height- and surface velocity profiles, which manifest themselves in a pronounced transition in the time evolution of the subphase morphology. With respect to both the spreading rates as well as the evolving morphology, excellent agreement between experimental and simulation results has been achieved.

Sunday, November 20, 2011 8:00AM - 9:57AM –
Session A18 Microfluids: General I 321

8:00AM A18.00001 Correlation-Force-Spectroscopy Rheometer[1] , MILAD RADIOM, BRIAN ROBBINS, CHRISTOPHER D.F. HONIG, JOHN Y. WALZ, MARK R. PAUL, WILLIAM A. DUCKER — We describe a new method, correlation force spectrometry, which characterizes fluids through measurement of the correlations between the thermally-stimulated vibrations of two closely spaced micrometer-scale cantilevers in fluid. We discuss applications to measurement of the rheological properties of complex fluids at high frequency and high spatial resolution. We measure a large range of frequencies (up to 1 MHz) and use very small sample volumes (μL) and demonstrate that the thermal noise in the cross correlation is much smaller than in the autocorrelation. Our experimental measurements of the equilibrium fluctuations in cantilever displacement for cantilever pairs immersed in a Newtonian fluid are described accurately using a theoretical approach based upon the fluctuation-dissipation theorem [Paul and Cross, Phys. Rev. Lett. 92, 235501 (2004)].

[1]The work described in this paper was funded by the National Science Foundation via Award Number CBET-0959228 and by Virginia Tech.

8:13AM A18.00002 High-throughput rheology in a microfluidic device[1] , ERIC FURST, KELLY SCHULTZ, University of Delaware, HYEJIN HAN, CHONGYOUP KIM, Korea University — High-throughput rheological measurements in a microfluidic device are demonstrated. A series of microrheology samples is generated as droplets in an immiscible spacer fluid using a microfluidic T-junction. The compositions of the sample droplets are continuously varied over a wide range. Rheology measurements are made in each droplet using multiple particle tracking microrheology. We review critical design and operating parameters, including the droplet size, flow rates and rapid fabrication methods. Validation experiments are performed by measuring the solution viscosity of glycerine and the biopolymer heparin as a function of concentration. Finally, an analysis of droplet mixing is performed in order to optimize the device performance. Overall, the combination of microrheology with microfluidics maximizes the number of rheological measurements while simultaneously minimizing the sample preparation time and amount of material, and should be particularly suited to the characterization of scarce or expensive materials.

[1]We acknowledge financial support from the NSF (CBET-0730292).

8:26AM A18.00003 High-Frequency Microrheology using a Cantilever Pair in Fluid[1] , BRIAN ROBBINS, MILAD RADIOM, CHRIS HONIG, JOHN WALZ, WILLIAM DUCKER, MARK PAUL, Virginia Tech — Measuring the rheological properties of small quantities of complex fluids at high frequencies remains an important challenge with many current technological applications. The field of two-point microrheology has made great strides yet most available approaches are subject to significant experimental and theoretical constraints regarding the accessible frequencies of measurement. In this talk, we explore the high-frequency motion of a pair of micron scale cantilevers in fluid to probe its rheological properties for a broad range of conditions. Using currently available cantilevers, it is possible to access MHz frequencies in a viscous fluid. We explore the motion of cantilever pairs that are either driven externally or by Brownian motion. We use the motion of an individual cantilever and the fluid-coupled motion of a cantilever pair to quantify the properties of the surrounding fluid. We focus here on analytical and numerical results for a range of Newtonian and power-law fluids and compare our findings directly with experimental measurements where possible.

[1]NSF Award CBET-0959228

8:39AM A18.00004 Rheology measurements using ciliated surfaces , ALEXANDER ALEXEEV, BRIAN T. JOHNSTON, YI YANG, Georgia Institute of Technology — We employ computational modeling to examine the utility of ciliated surfaces for measuring fluid viscosity. We consider a fluid-filled channel with a wall covered by compliant synthetic cilia that are arranged in a square pattern. The cilia can be actuated by a periodical force applied to their free ends and/or can be used to measure the bending moment at the points of cilium attachment to the wall. We show that the phase difference between the applied sinusoidal force and the bending moment at cilium root can be used to estimate fluid viscosity. We probe two types of ciliated surfaces. In the first scenario, alternating sensory and actuated cilia are arranged a chessboard pattern. In the second case, each cilium in the layer is both actuated and sensory. We compare these two layer arrangements in application to rheological measurements. In particular, we show that the phase signal is insensitive to the amplitude of applied force and layer density. Our results can be useful for designing sensory surfaces for microfluidic and biomedical applications.

8:52AM A18.00005 ABSTRACT WITHDRAWN –

9:05AM A18.00006 Photoinduced Contact Angle Hysteresis on a Single Microsphere[1] , SAMUEL ROSENTHAL, PATRICIA MCGUIGGAN, Johns Hopkins University — An atomic force microscope (AFM) is used to measure the meniscus force on individual microspheres as they contact and are retracted from an air/liquid interface. The glass microspheres, whose radii ranged from 20 to 50 micrometers, had organic or inorganic coatings on their surfaces. By exposing the microspheres to light, the contact angle and thus the meniscus force could be dramatically altered. The measured force-distance curves are fitted to macroscopic wetting theory. From these measurements, the contact angle, contact angle hysteresis, position of the contact line pinning, and surface tension were simultaneously determined.

[1]Supported by the 3M non-tenured Faculty Grant and the National Science Foundation under Grant No. 0709187.

9:18AM A18.00007 Effect of Partially Wetted Cavities on Superhydrophobic Friction Reduction , TAE JIN KIM, CARLOS HIDROVO, University of Texas at Austin — Superhydrophobicity refers to the condition of water-surface contact angle larger than 150°. Such microtextured surfaces with low sliding angles($<5°$) involve air pockets trapped underneath the water(Cassie state). This leads to a shear-free flow boundary condition and consequential reduction in frictional losses and pumping power requirements. We investigate the effects of partially wetted microtextured trenches on the friction reduction characteristics in microchannel flow. PDMS based superhydrophobic microchannels with sidewall trenches are used to visualize the water-air interface penetration and its effects on friction reduction. Theoretical models based on actual water layer measurements and shear free/no slip boundary conditions on the interface are used to calculate lower/upper bounds of the effective friction factor. These are compared to experimental values determined from pressure and flow rate data. Results suggest the existence of a "start-up" pressure required to overcome capacitance associated with surface tension effects and that friction reduction characteristics of the microchannels are unaffected by partial flooding of the trenches.

9:31AM A18.00008 Climbing of Micro-Droplets Along a Conical Fiber , ERQIANG LI, SIGURDUR THORODDSEN, KAUST — When a droplet is deposited onto a conical cylinder the spatial variation of capillary pressure makes it travel up the cone away from the tip. We use high-speed video imaging to observe the motion of micro-droplets along very small conical glass fibers. The results are compared to the pioneering work of Lorenceau & Quéré (2004) who used much larger cones. Our glass cones are made using a micro-pipette glass-puller, producing cone diameters down to about 5 micron at their tips. Numerous liquids, e.g. silicone oils, water and methanol were tested, to vary the viscosity over a range of 2000 and the surface tension over a factor of 3. Translations velocities from 0.1 to over 200 mm/s have been observed and the velocity follows capillary-viscous scaling. We find that there exists an optimum aspect ratio of the droplet shape for fastest motion.

9:44AM A18.00009 Simulation of microdroplet manipulation on flat surface actuated by wettability gradient using dissipative particle dynamics[1] , ZHEN LI, GUOHUI HU, ZHEWEI ZHOU, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University — A particle based mesoscopic methodology called dissipative particle dynamics (DPD) is utilized to simulate the manipulation (translation and coalescence) of micro-droplets in planar microfluidic chips with gradients of wettability. The three dimensional velocity field of a moving droplet propelled by wetting gradient is presented, which shows that the liquid spreads downwards onto substrate in the front area while it converges upwards in the back. The flow structure reveals that the drop moves in a combination of rolling and translation, and the rotation could be dominant when the contact angle is bigger than about 110°. The increasing of the gradient steepness and the size of droplet, as well as the magnitude of thermal fluctuation, is capable to significantly accelerate the movement for a sub-micrometer droplet. The liquid mixing in coalescence of two droplets is investigated as well. It is found that the process of mixing consists of two stages — a rapid active mixing through convective mass transfer and then a slow passive mixing through diffusion, which is consistent with previous experiment. Results also indicate that the thermal fluctuation is helpful to promote the coalescence and liquid mixing.

[1]The work was supported by the NSF of China (Grant No. 10872122).

Sunday, November 20, 2011 8:00AM - 10:10AM –
Session A19 Richtmeyer-Meshkov Instability I 322

8:00AM A19.00001 A pneumatic driver for shock wave production , MEGAN LEFTWICH, R. MEJILA-ALVAREZ, K. PRESTRIDGE, Los Alamos National Lab — We are presenting a novel technique to generate shock waves in shock tube experiments. Typically this is done with a high pressure driver section that is separated from the low pressure driven section by a physical membrane. The membrane is burst at a specific pressure and a shock wave is formed. This process limits the repetition of experiments, and membrane particles must be removed from the shock tube after each experiment. The driver presented here does not contain a membrane. Instead, it uses a series of high pressure chambers and fast-acting pistons to create the pressure jump between the high pressure driver section and low pressure driven section. The entire system is controlled remotely and requires no insertion or cleanup of membranes between experiments. The system is designed to achieve shock waves exceeding Mach 3 with air as the working fluid (higher Mach numbers can be generated with other working fluids). It will allow high repetition rates, even in challenging experimental environments (such as a vertical shock tube configuration). We present results from the initial characterization of this driver system.

8:13AM A19.00002 Vertical Shock Tube for simultaneous velocity and concentration measurements of Richtmyer-Meshkov Instabilities , R. MEJIA-ALVAREZ, K. PRESTRIDGE, M.C. LEFTWICH, Los Alamos National Laboratory — Most experimental studies on Richtmyer-Meshkov Instabilities (RMI) have been restricted to mixing layer growth and pointwise turbulence characterization. To date, the only exception to this trend encompasses simultaneous measurements of velocity and concentration via combined PIV and PLIF over a curtain of heavy gas with initial varicose perturbations.[1] Since no parallel of this work has been conducted on single interface configurations, the Extreme Fluids Team at Los Alamos National Laboratory has developed a new Vertical Shock Tube (VST) to carry out such measurements. When fully operational, this facility will allow simultaneous characterization of velocity and concentration fields at different stages of development of single-interface RMI flows. The extraction of turbulence statistics from velocity measurements will not only be instrumental in understanding the basic physics behind the single-interface RMI, but also the benchmark for RANS models and ILES. We present a description of the breath of functionality and diagnostic capabilities of LANL's new Vertical Shock Tube facility.

[1]Phys. Fluids 20, 124103 (2008)

8:26AM A19.00003 Mach number dependence of the Richtmyer-Meshkov instability with simultaneous density and velocity measurements , GREGORY ORLICZ, SRIDHAR BALASUBRAMANIAN, KATHY PRESTRIDGE, Los Alamos National Laboratory — Experiments are performed to study the effect of incident shock Mach number (M) on the development of the Richtmyer-Meshkov instability after a shock wave impulsively accelerates a varicose-perturbed, heavy-gas curtain (air-SF_6-air). Incident shock strength is varied within the weak shock regime (M $\leq$ 1.5), and the resulting instability and subsequent fluid mixing is measured using simultaneous Planar Laser-Induced Fluorescence (PLIF) and Particle Image Velocimetry (PIV). We investigate the mechanisms that drive the instability, at both large and small scales, by examining the time evolution of simultaneous, 2-D density and vorticity fields for each Mach number. Mixing layer width, vorticity, circulation, velocity fluctuations, turbulent kinetic energy, and the density self-correlation parameter are examined as a function of time. These quantities are also examined versus time scaled with the convection velocity of the mixing layer, showing that the rate of change of several of these quantities is independent of Mach number.

8:39AM A19.00004 Single and multi-mode initial condition influence on turbulent mixing in Richtmyer-Meshkov flows[1] , SRIDHAR BALASUBRAMANIAN, GREG ORLICZ, KATHY PRESTRIDGE, Los Alamos National Laboratory, EXTREME FLUIDS TEAM — Experimental evidence is needed to verify and validate the numerical hypothesis that shock-driven flows retain the memory of initial conditions. We present the results of an experimental study to understand the influence of initial condition parameters, namely amplitude and wavelength of perturbations, on mixing and transition in Richtmyer Meshkov flows. Single and multi-mode membrane-free initial conditions in form of a gas curtain (*air-*SF_6*-air*) at Mach number $M = 1.2$ and Atwood number, A=0.67 was used. The evolution of instability is captured using high resolution simultaneous PLIF and PIV. Statistics such as mixing widths, density self-correlation parameter, turbulent kinetic energy, turbulent Reynolds number, and variances of velocity fluctuations were measured to quantify the amount of mixing. Some of these statistics were found to be in disagreement with the linear theory. Based on the results, a correlation between mixing at late times and initial condition parameters is found.

[1]This work is supported by Los Alamos Laboratory Directed Research & Development Program.

8:52AM A19.00005 The Richtmyer-Meshkov Instability of a New Type of Broadband Initial Condition , CHRISTOPHER WEBER, NICHOLAS HAEHN, JASON OAKLEY, DAVID ROTHAMER, RICCARDO BONAZZA, University of Wisconsin-Madison — The Richtmyer-Meshkov instability is experimentally investigated using a broadband initial condition imposed on an interface of helium+acetone over argon. The initial condition is created, first by setting up a gravitationally stable stagnation plane, and then injecting the gases horizontally at the interface to create a shear layer. The perturbations along the shear layer create a statistically repeatable broadband initial condition. The interface is accelerated by a M = 1.6 and M = 2.1 planar shock wave, inducing a growth of the interface perturbations. The development of this turbulent mixing layer is investigated using PLIF.

9:05AM A19.00006 Planar Shock Acceleration of a Droplet-Seeded Gas Jet: Three-Dimensional Features[1] , PETER VOROBIEFF, JOSEPH CONROY, MICHAEL ANDERSON, ROSS WHITE, C. RANDALL TRUMAN, The University of New Mexico, SANJAY KUMAR, University of Texas - Brownsville — When a planar shock wave generated in a shock tube accelerates a nominally two-dimensional density interface, the large-scale flow structure is usually regarded as quasi-two-dimensional. We examine the limitations of this assumption due to interaction with wall boundary layers, growth of three-dimensional instabilities, and other factors. The initial conditions are produced by a laminar cylindrical jet of gas vertically injected into the test section of the shock tube. Flow visualization images in several planes intersecting the flow reveal a non-trivial three-dimensional structure. Experiments are conducted both for "classical" Richtmyer-Meshkov instability and for its multiphase analog induced by particle seeding.

[1]This work is supported by NNSA through DOE Grant DE-PS52-08NA28920 and by DTRA via awards HDTRA1-07-1-0036 and HDTRA1-08-1-0053.

9:18AM A19.00007 Modeling a Shock-Accelerated Fluid - Multiphase Fluid Interface , MICHAEL ANDERSON, PETER VOROBIEFF, ROSS WHITE, JOSEPH CONROY, C. RANDALL TRUMAN, The University of New Mexico, SANJAY KUMAR, University of Texas - Brownsville — The hydrocode SHAMRC has been used in the past to study the formation and growth of the Richtmyer-Meshkov Instability (RMI). While RMI involves impulsively accelerating two continuous fluids of differing densities, a similar class of instabilities has recently been described for multiphase flow. In this scenario, a shock wave passes through a region containing ambient air seeded with particles which have a non-trivial mass and density much greater than that of the surrounding and embedding fluid. In this scenario, no baroclinic vorticity is generated due to the lack of a fluid-fluid density interface. After the shock passage, the particles or droplets lag behind the surrounding gas. Momentum exchange between the embedded phase and the embedding phase leads to non-uniform local equilibrium velocity distribution, and thus to shear and vortex formation. As the primary mechanism for this instability formation is momentum transfer via drag, the morphology of the instability is strongly dependent of the sizes of the particles in the initial conditions. The simulations described here attempt to model the effects of changing the particle size on the morphology and growth rate of this instability.

9:31AM A19.00008 Shock-initiated Combustion with New Insights into the Nature of the Shock-focusing Phenomenon[1] , NICHOLAS HAEHN, CHRISTOPHER WEBER, JASON OAKLEY, DAVID ROTHAMER, University of Wisconsin-Madison, DEVESH RANJAN, Texas A&M University, RICCARDO BONAZZA, University of Wisconsin-Madison — Shock-focusing that results from the interaction of a planar shock wave with a spherical density inhomogeneity is used to ignite a reactive mixture of gases. Due to the singular nature of this process, the task of quantifying the effect of the shock-focusing is challenging from a numerical and analytical point of view. As such, there is a lack of understanding regarding the thermodynamic conditions that are achieved during the shock-focusing process. These conditions, and this process in general, are important to a wide range of disciplines, including inertial confinement fusion, astrophysics, and supersonic combustion. A bubble is prepared using a stoichiometric mixture of fuel and oxidizer and diluted with Xe, which increases the overall density of the mixture. The experiments are performed in the Wisconsin Shock Tube Laboratory (WiSTL) in a 9.2 m vertical shock tube with a 25.4 cm ×g5.4 cm square cross-section. The bubble is accelerated by a planar shock wave ($2.0 < M < 2.8$). Planar Mie scattering and chemiluminescence are used simultaneously to visualize the bubble morphology and combustion characteristics. In turn, the combustion can be used as a diagnostic to assess the conditions that exist near the shock-focusing region.

[1]DOE/NNSA Aware number DE-NA0000629 and NSF Award 0827285.

9:44AM A19.00009 Oblique Shock Interaction with a Gas Cylinder[1] , ROSS WHITE, JOSEPH CONROY, MICHAEL ANDERSON, PETER VOROBIEFF, C. RANDALL TRUMAN, The University of New Mexico, SANJAY KUMAR, University of Texas - Brownsville — In the majority of shock-tube studies of Richtmyer-Meshkov instability, when a planar shock interacts with a perturbed density interface, the orientation of the interface plane or the largest interface feature (e.g., axis of a gaseous column) is parallel to the plane of the shock. Here we experimentally study the flow developing after an interaction of an *oblique* shock wave with a gravity-stabilized cylindrical heavy gas (SF_6) column surrounded by less dense gas (air). To introduce an oblique angle into the initial conditions, we tilt the shock tube to an angle of 15° with respect to the horizontal. Flow visualization in several planes is conducted to highlight the differences between the features characterizing planar and oblique shock-cylinder interaction. Several flow structures peculiar to oblique interaction appear to exist over a range of Mach numbers from 1.2 to 2.1.

[1]This work is supported by NNSA through DOE Grant DE-PS52-08NA28920 and by DTRA via awards HDTRA1-07-1-0036 and HDTRA1-08-1-0053.

9:57AM A19.00010 The Oscillation and Rupture of a Water-filled Balloon , HUGH LUND, STUART DALZIEL, DAMTP, University of Cambridge — Experimental observations of the impact of a water-filled balloon on a rigid surface have shown that capillary-like waves form on the membrane, with the tension in the membrane as the restoring force. If the membrane ruptures during the impact, two forms of instability occur on the air/water interface. First, the rapid retraction of the membrane creates a small-scale shear instability. Second, larger scale growth of the interfacial amplitude occurs, as the restoring force for the capillary-like waves is lost while the kinetic energy within the water remains. A water-filled balloon that is held and forcibly oscillated then ruptured with a sharp object displays the same three phenomena: capillary-like waves, shear instability and larger-scale interfacial growth. In air, the large-scale growth of the interfacial amplitude becomes asymmetric, leading to the formation of so-called bubbles of the air phase and spikes of the water phase. For such a balloon held underwater, the interfacial growth is symmetric. In this paper, we present examples of the phenomena described above. Further, we explain why the late-time growth of the interfacial amplitude is a manifestation of the Richtmyer-Meshkov instability. Unlike the classical instability, growth may occur when there is no density difference across the interface. At late-time, measurements of the displacement of the maximum amplitude of the interface suggest a power law of the form t^{θ}, where t is time and θ is around $2/3$.

Sunday, November 20, 2011 8:00AM - 10:10AM –
Session A20 Interfacial/Thin Film Instability I 323

8:00AM A20.00001 Predicting Stability of Air–Water Interface on Superhydrophobic Surfaces[1] , B. EMAMI, H. VAHEDI TAFRESHI, M. GAD-EL-HAK, G.C. TEPPER, Virginia Commonwealth University — In this work, two different methodologies for predicting the stability of the air-water interface on submerged superhydrophobic surfaces are presented. The first method is an analytical approach developed by balancing the hydrostatic pressure with the capillary forces over the interface, and results in a second-order partial differential equation. The solution to this equation provides the 3-D interface shape and the critical pressure beyond which the superhydrophobic surface departs from the Cassie state. The second method presented here is an approximate numerical technique based on the so called Full Morphology method in which the Young–Laplace equation is used to relate a capillary pressure to the most constricted opening of the pore space between the peaks of the surface roughness. Predictions of the methods presented in this study are compared with the available studies in the literature (*Applied Physics Letters* **98**:20, 203106, 2011).

[1]Financial support from DARPA, contract number W91CRB-10-1-0003, is acknowledged.

8:13AM A20.00002 Three-dimensional convective and absolute instabilities in pressure-driven two-layer channel flow , KIRTI SAHU, Department of Chemical Engineering, Indian Institute of Technology Hyderabad, Yeddumailaram 502 205, Andhra Pradesh, India, OMAR MATAR, Department of Chemical Engineering, Imperial College London, South Kensington Campus, London, SW7 2AZ, UK — A generalized linear stability analysis of three-dimensional disturbance in a pressure-driven two-layer channel flow, focusing on the range of parameters for which Squire's theorem does not exist is considered. Three-dimensional linear stability equations, in which both the spatial wavenumber and temporal frequency are complex, are derived and solved using an efficient spectral collocation method. A Briggs-type analysis is then carried out to delineate the boundaries between convective and absolute instabilities in m-Re space. We find that although three-dimensional disturbances are temporally more unstable than the two-dimensional disturbances, absolute modes of instability are most unstable for two-dimensional disturbances. An energy "budget" analysis also shows that the most dangerous modes are "interfacial" ones.

8:26AM A20.00003 Diffuse interface approach to rotating Hele-Shaw flows , JOSE MIRANDA, Dept. Fisica, UFPE - Brazil, CHING-YAO CHEN, YU-SHENG HUANG, Dept. Mech. Engineering, National Chiao Tung University, Taiwan — When two fluids of different densities move in a rotating Hele-Shaw cell, the interface between them becomes centrifugally unstable and deforms. Depending on the viscosity contrast of the system distinct types of complex patterns arise at the fluid-fluid boundary. Deformations can also induce the emergence of interfacial singularities and topological changes such as droplet pinch-off and self-intersection. We present numerical simulations based on a diffuse interface model for this particular two-phase displacement that capture a variety of pattern forming behaviors. This is implemented by employing a Boussinesq Hele-Shaw-Cahn-Hilliard approach, considering the whole range of possible values for the viscosity contrast, and by including inertial effects due to the Coriolis force. The role played by these two physical contributions on the development of interface singularities is illustrated and discussed.

8:39AM A20.00004 Solitary waves running down a vertically falling film: low-dimensional models[1] , SYMPHONY CHAKRABORTY, CHRISTIAN RUYER-QUIL, UPMC, Lab. FAST, campus universitaire, 91405 Orsay, France, PHUC KHANH NGUYEN, VASILIS BONTOZOGLOU, Dept. ME, UTh, Volos, 38334, Greece — We consider the wavy regime of vertically falling fluid film. Derivation of LDM for this flow, based on the LW expension have a long history(see review in[1]). A crucial test of such models is the correct prediction of the properties of SW as a function of the distance from the instability threshold. The latter is usually quantified in terms of the reduced Re, $\delta = 3^{\frac{4}{3}} Re^{\frac{11}{9}} Ka^{\frac{-1}{3}}$. Though most models predict similar behavior in the drag-gravity regime($\delta \ll 1$), they exhibit large differences from each other in the drag-inertia regime($\delta \gg 1$). Characteristics of SW solutions to available LDM are shown and contrasted with DNS results. The best agreement with DNS is found with the 4-equation model derived in[2] and the convergence rate of the wave speed to the asymptotic limit c_∞ at $\delta \to \infty$ is affected by viscous diffusion terms and is governed by the Re as $|c - c_\infty| \propto 1/Re^2$. The asymptotic behavior of the speed, amplitude, lengths of the wave-tail and front capillary ripples are discussed. References: [1] R.V.Craster, O.K.Matar. Reviews of modern physics 81 (2009). [2] C.Ruyer-Quil, P.Manneville. Eur.Phys.J. 15 357-369 (2000).

[1]Research partially supported by EU/FP7-ITN-MULTIFLOW.

8:52AM A20.00005 Three-dimensional instabilities of miscible fingers in a Hele-Shaw cell , RAFAEL OLIVEIRA, UCSB, FELIX HEUSSLER, MICHAEL JOHN, ECKART MEIBURG, UCSB — We perform three-dimensional DNS simulations of the transient, variable viscosity Boussinesq Navier-Stokes equations, coupled to a convection-diffusion equation for a concentration field, to simulate miscible viscous fingers in Hele-Shaw cells. The three-dimensional problem allows for new instabilities and patterns that cannot be captured by traditional gap-averaged modeling. For constant density displacements, we find that a streamwise vorticity quadrupole forms that induces fluid transport from the walls of the cell to its center, thereby leading to a new hydrodynamic instability, termed "inner splitting." If gravity is included, the nature of the two-dimensional base flow and its subsequent instability changes dramatically. The interaction between Saffman-Taylor and Rayleigh-Taylor instabilities can lead to additional splitting events, and it can significantly enhance the mixing rates of the two fluids, thereby altering the overall displacement efficiency. This work is supported by NSF, and a CAPES/Fulbright fellowship.

9:05AM A20.00006 Thin film drainage of a deformable droplet moving toward a wall with finite inertia

, SHAOPING QUAN, Institute of High Performance Computing — Direct numerical simulations of a deformable droplet approaching toward a solid wall through another fluid are performed by solving the full Navier-Stokes equations using a finite volume/ moving mesh interface tracking method with high fidelity. Cases with Reynolds numbers of 25 and 50 and capillary numbers of 5×10^{-3} and 1×10^{-2} are simulated for both head-on and oblique approaching scenarios. The front head of the droplet is flattened as the droplet nearly touches the wall, and a dimpled thin film is observed. Because of the great viscous forces of the flow inside the thin film, the droplet slows down dramatically which leads to a significant increase of the drag force. An asymmetric thin film is observed for the oblique approaching. The numerical prediction on the central separation at which a dimple is formed agrees fairly well with previous analysis based on the lubrication theory. The simulated thinning rate is slower than the rate predicted by previous approximate models. The differences are mainly due to the finite Reynolds number of the simulated cases.

9:18AM A20.00007 Contact line induced instabilities for thin fluid films[1]

, TE-SHENG LIN, LOU KONDIC, New Jersey Institute of Technology — We study contact line induced instabilities for thin film of complete and partially wetting fluids spreading down an inclined plane with inclination angle ranging from 0 to pi. It is found that a contact line may lead to free surface instability without any additional perturbations. We investigate the effect of both inclination angle and of contact angle on surface and transverse (fingering) instabilities, as well as on dewetting process.

[1]Partially supported by NSF Grant No. DMS-0908158.

9:31AM A20.00008 The dynamics of foams with mobile interfaces[1]

, MICHAEL B. GRATTON, STEPHEN H. DAVIS, Northwestern University — Using a novel technique for resolving nearly singular integrals, we investigate the dynamics of two-dimensional foams with mobile interfaces and an incompressible, inviscid gas phase by a boundary integral method. For foams with small liquid fractions ($\leq 5\%$), we observe node motion, lamellar bending, drainage, and T1 transitions. Node motion occurs on the fastest timescale and is well-described by considering only the surface forces on each Plateau border. The lamellar bending is characterized by viscida theory, but the drainage occurs at a different rate than predicted by the asymptotic theory for the examples studied. Topological transitions occur on a timescale intermediate to Plateau border rearrangement and drainage.

[1]Supported by NSF grants DMS-0636574 and CMMI-0826703.

9:44AM A20.00009 Optical interference effect on Marangoni instability of irradiated thin liquid films

, FUMIHIRO SAEKI, SHIGEHISA FUKUI, HIROSHIGE MATSUOKA, Tottori University — The Marangoni instability of thin liquid films on solid substrates induced by irradiative heating is investigated within the framework of the long-wave approximation. The energy transfer that includes the energy absorption and reflection is taken into account. In order to examine the optical interference effects on the instability, focus is placed on a transparent film/absorbable substrate system irradiated by a monochromatic wave with laterally uniform intensity distribution. In such a case, the energy reflectance varies periodically with the film thickness due to optical interference. Numerical simulation results show that the stability of the film depends on the first derivative of the energy reflectance with respect to the film thickness at a reference point, and the resultant surface patterns differ depending on the reference thickness and initial perturbation.

9:57AM A20.00010 Numerical simulations of two-fluid boundary layers beneath free-stream turbulence

, SEO YOON JUNG, TAMER ZAKI, Imperial College London — In two-fluid boundary layers, a wall-film is sheared by an external stream with different density and viscosity. As a result, the flow becomes prone to both shear and interfacial instabilities. In this study, the evolution of two-fluid boundary layers beneath free-stream vortical forcing is investigated using DNS. The simulations employ a conservative level-set technique in conjunction with a ghost fluid approach in order to capture a sharp interface. The wall film is less viscous than the outer flow, and its thickness is 10% of that of the boundary layer at the inlet. The choice of viscosity ratio influences the spatial development of disturbances within the boundary layer. The spatial growth of instabilities is examined into the non-linear regime, which includes the region of breakdown to turbulence. We demonstrate that, at moderate levels of free-stream turbulence intensities, appropriate choice of the viscosity ratio can yield considerable transition delay.

Sunday, November 20, 2011 8:00AM - 10:10AM –

Session A21 Vortex Dynamics I 324-325

8:00AM A21.00001 Effects of formation time and flexibility on the starting and stopping vortices in piston-cylinder devices[1]

, MEHDI SAADAT, HOSSEIN HAJ-HARIRI, University of Virginia — A computational study has been conducted to address the formation of vortex rings ejected from a weakly flexible nozzle at the end of a piston-cylinder device. The first observation is that, regardless of flexibility, for small enough duration of a push on the piston, the negatively-signed induced vorticity on the outside of the cylinder merges with the stopping vortex and spills into the core and pairs up with the primary (starting) vortex. The second observation is that the flexibility of the exit nozzle affects the behavior of the vortex by changing the effective diameter of the exit. Also, the snapping back of the nozzle strengthens this interaction. To model the flexibility, two approaches have been used in the study. First, the nozzle is modeled using a torsional-spring-mass-rigid system which is allowed to rotate about a hinge and its rotation is coupled with the flow solver. Second, the nozzle is considered as an elastic material and its deflection is solved using a FEM solver. It is hypothesized that the presence of flexibility in the model increases the time scale as well as the thrust as compared with results from rigid nozzle models. A study is conducted to find the highest thrust generated versus the flexural parameters.

[1]supported by ONR MURI

8:13AM A21.00002 Computational Analysis of Vortex Formation Over a Flat Plate with Gradient-Based Slip Condition

, JOHN PALMORE, AMY LANG, MUHAMMAD SHARIF, University of Alabama — A thorough understanding of small scale aerodynamics is important for the design of micro air vehicles. Since they fly in the same Re regime as that of insects, these animals can provide biologically inspired designs. Butterflies have small scales (on the order of 100 microns in length) that line the surface of their wings; these scales can affect the slip condition over their wings, altering vortex formation and possibly leading to improved flight characteristics. In particular, it is believed that the scales impose a preferential direction for flow over the wing. To test this hypothesis, 2D flow over an infinitely thin flat plate was studied using the CFD software FLUENT. The slip condition was specified by defining an imposed velocity gradient on the plate's surface. Differences in vortex growth and formation are discussed with the goal of ascertaining whether the existence of a preferential flow direction leads to improved flight performance.

8:26AM A21.00003 Optimal energy harvesting from long cables in VIV , CLEMENT GROUTHIER, SEBASTIEN MICHELIN, EMMANUEL DE LANGRE, Ecole Polytechnique - LadHyX — Vortex-induced vibrations (VIV) of flexible structures in steady flows may result in large amplitude self-sustained oscillations of the solid, making it an attractive mechanism for energy harvesting. Here, we investigate the possibility of harvesting energy using long tensioned cables in VIV with localized energy extraction. The non-linear fluid-solid dynamics is described using a classical Van der Pol wake oscillator and the energy harvesting device is represented by a discrete damping distribution (dashpots). The cable extracts energy from the flow along its entire length, but it is also responsible for the transport of the harvested energy toward the dashpots. Focusing on the permanent saturated regime, we study this energy transfer along the cable, and determine the optimal damping distribution that maximizes the harvested energy.

8:39AM A21.00004 Phase drift between the in-line and cross-flow vortex-induced vibrations of a long flexible cylinder in shear flow , REMI BOURGUET, IMFT, GEORGE KARNIADAKIS, Brown University, MICHAEL TRIANTAFYLLOU, MIT — Long flexible structures with bluff cross-sections placed in cross-flow exhibit vortex- induced vibrations. The in-line and cross-flow vibrations generally occur with a frequency ratio of two leading to figure-eight orbits in the plane perpendicular to the span. The structure is excited by the flow under a lock-in condition defined as the synchronization between vortex shedding and body displacement. Previously, we have shown that, in sheared current, the region of lock-in is characterized by trajectories where the structure moves upstream at the extremes of the cross-flow motion. Recent experimental and numerical studies have emphasized a phenomenon of phase drift between in-line and cross-flow vibrations dominated by traveling waves, that induces a continuous change in the shape and orientation of the flexible body orbit along the span. In the present work, we elucidate the link between this phenomenon and the departure from a ratio of two between the in-line and cross-flow excited structural wavenumbers and we show, by means of numerical simulation, that the effective added mass in each direction controls the phase drift.

8:52AM A21.00005 Numerical simulations of flow past flat plates , LING XU, Univeristy of New Mexico, MONIKA NITSCHE, University of New Mexico — The starting vortex in flow past a finite and a semi-infinite plate is studied numerically, using a semi-Lagrangian, compact finite difference method with an implicit time-stepping scheme. The results are compared with those existing in the literature for driven cavity flow and flow past plates. The effects of viscosity on the core trajectory and the shed circulation are investigated. Furthermore, an alternative vortex method is explored which has more flexibility in terms of the fluid domain, and is potentially much faster, provided a fast summation algorithm is used.

9:05AM A21.00006 Unsteady vortical structures in porous media flows[1] , JUSTIN FINN, SOURABH APTE, BRIAN WOOD, Oregon State University — The pore scale character of moderate Reynolds number, inertial flow through mono-disperse packed beds of spheres is examined using numerical experiments. Direct numerical simulations are performed for flow through (i) a periodic, $3 \times 3 \times 6$ simple cubic arrangement at $Re_p = 529$, and (ii) a realistic randomly packed tube containing 326 spheres with $d_{tube}/d_{sp} = 5.96$ at $Re_p = 600$. At these Reynolds numbers, unsteady vortical regions are dominant features at the pore scale, and can have a profound effect on permeability and dispersion properties at the macro-scale. Despite similar Reynolds numbers and mean void fractions, the vortical structures observed in these two flows are remarkably different. The flow through the arranged packing is characterized by spatially and temporally periodic vortex-ring like structures, while the flow through the random packing contains many elongated helical vortices and a wider spectrum of space and time scales. The sensitive dependence of flow length and time scales and the local pore geometry is investigated using the DNS data.

[1]Funding: NSF project #0933857, Inertial Effects in Flow Through Porous Media

9:18AM A21.00007 POD-based Reduced-order Model for Forced Motion and Vortex-Induced Vibration of a Circular Cylinder , ARNE J. PEARLSTEIN, RAVI KUMAR R. TUMKUR, GUY BEN-DOV[1], RAMON CALDERER, ARIF MASUD, LAWRENCE A. BERGMAN, ALEXANDER F. VAKAKIS, University of Illinois at Urbana-Champaign — For flow past a cylinder either held fixed, in prescribed harmonic motion perpendicular to the mean flow, or mounted on a linear spring and undergoing vortex-induced vibration (VIV) perpendicular to the mean flow, a finite-element method (FEM) is used to compute high-fidelity solutions of the Navier-Stokes equations at $Re = 100$ (based on diameter). By means of proper orthogonal decomposition, the computed velocity field is used to construct a set of orthogonal vector-valued velocity modes. The pressure field is decomposed into a set of nonorthogonal modes. An approximate reduced-order model (ROM) is obtained by projecting the Navier-Stokes equations onto these modes. To stabilize the resulting ordinary differential equation system, we add a "shift mode" calculated using the steady symmetric solution for a stationary cylinder. For all three cases (stationary cylinder, prescribed motion, and VIV), the ROM gives velocity and pressure fields in good agreement with the high-fidelity FEM results.

[1]Present address: Université de Poitiers, Poitiers, France

9:31AM A21.00008 Passive Suppression of Vortex-Induced Vibration of a Cylinder at $Re = 100$, RAVI KUMAR R. TUMKUR, RAMON CALDERER, ARIF MASUD, LAWRENCE A. BERGMAN, ARNE J. PEARLSTEIN, ALEXANDER F. VAKAKIS, University of Illinois at Urbana-Champaign — For a Reynolds number (*Re*) based on cylinder diameter of 100 and a ratio of cylinder density to fluid density of 10, we investigate the use of an essentially nonlinear approach to passive suppression of vortex-induced vibration (VIV) of a rigid circular cylinder restrained by a linear spring, and constrained to move perpendicular to the mean flow. The variational multiscale residual-based stabilized finite-element method used to compute approximate solutions of the incompressible Navier-Stokes equations about the moving cylinder is coupled to a simple model of a "nonlinear energy sink" (NES), an essentially nonlinear oscillator consisting of a mass, a linear damper, and a strongly nonlinear spring. The NES promotes nearly one-way transfer of energy from the primary structure (the cylinder) to itself, resulting in reduction of the amplitude of the limit cycle oscillation by as much as 75%, depending on the parameters characterizing the NES. Various mechanisms of VIV suppression by the NES are discussed, along with results showing the effectiveness, over a range of *Re*, of passive suppression using an NES whose parameters were selected to work well at $Re = 100$.

9:44AM A21.00009 Flow around a cylinder fitted with strakes , JULIO R. MENEGHINI, RAFAEL S. GIORIA, NDF, Dept. Mech. Eng., Escola Politecnica, University of Sao Paulo — In this paper, we investigate numerically the mechanism of suppression of vortex-induced vibration by using helical strakes. Although the strakes effectiveness has been assessed, the suppression mechanism is yet not well established. Most works point out that the main aspect on the suppression is the decrease in the vortex shedding correlation along the span. The flow in the spanwise direction induced by the shape of the strakes is also another reason for suppression of vortex-induced vibration. Computational simulation of the flow around a fixed circular cylinder fitted with strakes are carried out for Reynolds number 1,000 and 10,000 in order to ratify that the main aspect on the suppression mechanism is the decrease in the vortex shedding correlation along the span. To support the latter allegation, we employ a spatial correlation of the mean velocity fields, and we also investigate the instantaneous resulting fluid forces on different sections along the body in comparison to experimental measurements. We also observe that the vortex wake is formed farther downstream and we can identify this as a secondary mechanism of vortex-induced vibrations suppression. A comparison of velocity fields from particle image velocimetry with the simulation results is presented in order to support the simulation analysis.

9:57AM A21.00010 ABSTRACT WITHDRAWN —

Sunday, November 20, 2011 8:00AM - 9:31AM –
Session A23 Rarefied Gases and DSMC 326

8:00AM A23.00001 A thermally driven ratchet for the separation of gas molecules , STEFFEN HARDT, Center of Smart Interfaces, TU Darmstadt, SUDARSHAN TIWARI, AXEL KLAR, Fachbereich Mathematik, TU Kaiserslautern — If a gas is confined between two suitably structured surfaces, the application of a temperature gradient between the surfaces gives rise to nontrivial phase-space distributions inside the gas if the Knudsen number is larger than one. In such non-equilibrium situations a nonzero momentum and mass flux can be created in a direction parallel to the surfaces. We study a situation of comparatively large molecules of low concentration moving in a background gas of small molecules. It is assumed that the Knudsen number of the small molecules is much larger than one. Therefore, the small molecules dominantly collide with the wall boundaries, whereas the large molecules undergo collisions with the walls as well as the small molecules. Such a scenario is analyzed using Monte-Carlo simulation techniques based on a hard-sphere collision model. It is shown that while the small molecules show no net motion, the large molecules are driven parallel to the surfaces with a velocity that depends on their size. Larger molecules are transported faster than smaller ones. In consequence we have demonstrated a novel scheme of gas dynamics that may find applications in the size separation of molecules.

8:13AM A23.00002 Quadrature–based moment closures for non–equilibrium flows: hard–spheres collisions and approach to equilibrium , MATTEO ICARDI, PIETRO ASINARI, DANIELE MARCHISIO, Politecnico di Torino, SALVADOR IZQUIERDO, Instituto Tecnológico de Aragón and Universidad de Zaragoza, RODNEY FOX, Iowa State University — Recently the Quadrature Method of Moments (QMOM) has been extended to solve several kinetic equations, in particular for gas–particle flows and rarefied gases. This method is usually coupled with simplified linear models for particle collisions. In this work QMOM is tested as a closure for the dynamics of high–order moments with a more realistic collision model namely the hard–spheres model in the Homogeneous Isotropic Boltzmann Equation. The behavior of QMOM far away and approaching the equilibrium is studied. Results are compared to other techniques such as the Lattice–Boltzmann (LBM) and the Grad's expansion (GM) methods. Comparison with a more accurate and computationally expensive model, based on the Discrete Velocity Method (DVM), is also carried out. Our results show that QMOM describes very well the evolution when it is far away from equilibrium, without the drawbacks of the GM and LBM or the computational costs of DVM but it is not able to accurately reproduce the equilibrium and the dynamics close to it. Corrections to cure this behavior are proposed and tested.

8:26AM A23.00003 DSMC Simulations Assessing the ES-BGK Kinetic Model for Gas-Phase Transport between Parallel Walls , M.A. GALLIS, J.R. TORCZYNSKI, Sandia National Laboratories — Bird's Direct Simulation Monte Carlo (DSMC) method is used to simulate gas-phase diffusive transport at near-continuum conditions. The molecules collide using either the Boltzmann collision term or the ellipsoidal-statistical Bhatnagar-Gross-Krook (ES-BGK) kinetic model. Momentum, heat, and mass transport between parallel walls (i.e., Couette, Fourier, and Fickian flows) are investigated. The ES-BGK model produces values of the viscosity and the thermal conductivity outside the Knudsen layers that agree closely with the corresponding values from the Boltzmann collision term (also implemented in DSMC). However, the ES-BGK model produces less accurate values for the mass self-diffusivity, with a modest difference for the Maxwell interaction but a large difference for the hard-sphere interaction. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

8:39AM A23.00004 An Expression for the Gas Mass Flow Rate through a Tube from Free-Molecular to Continuum Conditions , J.R. TORCZYNSKI, M.A. GALLIS, Sandia National Laboratories — An expression for the steady isothermal gas mass flow rate through a long thin tube from free-molecular to continuum conditions with arbitrary accommodation is developed. This expression is based on the Navier-Stokes equations and a slip boundary condition developed with the philosophy that the mass flow rate is more important than the velocity field. Its form permits integration along the tube to obtain a closed-form expression. This expression contains three coefficients. The first and the second are known from free-molecular and near-continuum flow. The third is determined from Direct Simulation Monte Carlo (DSMC) simulations for flows in the transitional regime. A similar expression is developed for rectangular channels. These expressions agree well with recent experiments measuring mass flow rates through microscale tubes and channels. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

8:52AM A23.00005 Characteristics Based Boundary Conditions for 3D Unstructured DSMC Simulations , SERGEY AVERKIN, NIKOLAOS GATSONIS — The theoretical and algorithmic framework of a characteristic-based method for subsonic boundary conditions suitable to the Direct Simulation Monte Carlo (DSMC) is presented. This method is based on the local one dimensional inviscid (LODI) formulation used in compressible (continuous) flow computations. The method is implemented in an unstructured 3D DSMC (U3DSMC) code and is supplemented with a neighboring-cell sampling approach and a time-average smoothing technique to speed convergence and reduce fluctuations. Simulations of a pressure-driven subsonic flow in a circular tube are used for verification and validation of the boundary condition method. The length and the radius of the tube are 0.02m and 0.001m respectively. The inlet surface is specified with a fixed pressure 300Pa and a number density 7.2464e+22 m^{-3} and the outlet surface is specified with a fixed pressure 150Pa. The corresponding Knudsen number at the inlet is 0.1786 and at the exit 0.3572 covering the transitional rarefied flow regime. The sidewall is modeled as a diffuse surface with full accommodation to a temperature of 300K. The average inlet Mach number is 0.156. Local error estimates are compared with theoretical predictions. The numerical results are in good agreement with theoretical and experimental results.

9:05AM A23.00006 Effect of weak gravitation on the plane Poiseuille flow of a highly rarefied gas , TOSHIYUKI DOI, Tottori University — Plane Poiseuille flow of a highly rarefied gas that flows horizontally in the presence of weak gravitation is studied based on the Boltzmann equation for a hard sphere molecular gas. The behavior of the solution in the regime of large mean free path and small strength of gravity is studied numerically based on the one-dimensional Boltzmann equation derived by means of an asymptotic analysis for a slow variation in the flow direction. It is clarified that the effect of weak gravity on the flow is not negligible when the mean free path is so large that it is comparable to the maximum range of the parabolic molecular path within the channel. When the mean free path is much larger than this range, the effect of gravity that makes the molecules fall plays the dominant role in determining the distribution function, and thus the over-concentration in the distribution function as well as the flow velocity does not grow further even if the mean free path is increased.

9:18AM A23.00007 Continuous Stochastic Equations for Diatomic Rarefied Gas Flows , HOSSEIN GORJI, PATRICK JENNY, Institute of Fluid Dynamics, ETH Zurich — In this talk, we postulate a non-linear Fokker-Planck model for simulations of the rarefied gas flows whereas the gas molecules would possess the internal degrees of freedom (DoF). The main motivation of the proposed model is the computational efficiency which is obtained due to the fact that in this solution algorithm no collisions between notional particles have to be calculated. The model equation is based on the Fokker-Planck approximation of the Boltzmann equation which has already been used for the motion of monatomic gas molecules by authors. However, a new set of stochastic differential equations (SDEs) is proposed for internal modes of the diatomic molecules. Based on the proposed model the heat conductivity of nitrogen is calculated for the range of temperatures from 200 to 800 K and it is shown that excellent agreement with regard to experimental results is gained.

Sunday, November 20, 2011 8:00AM - 10:10AM –
Session A24 Suspensions I 327

8:00AM A24.00001 The Role of the Stokes Number, the Capillary Number, and the Centrifugal Number in Oblique Wetted Particle Collisions , CARLY DONAHUE, ROBERT DAVIS, CHRISTINE HRENYA, University of Colorado, Boulder — A plethora of systems that contain solid particles coated in a liquid layer are found in nature (e.g. avalanches, pollution) and in industry (e.g. granulation, particle filtration). Previous micro-level studies of wetted collisions have been limited to normal particle-particle collisions and normal and oblique particle-wall collisions. However, for a complete understanding, a study of oblique particle-particle collisions is crucial. In the work presented here, a combination of experiments and theory are used to identify the roles of capillary and centrifugal forces in these collisions at low Reynolds number. Surprisingly, even when the capillary number is high (ratio of lubrication to capillary forces), capillary forces play an essential role in agglomeration vs. de-agglomeration. Moreover, another dimensionless number is identified to characterize the relative importance of the centrifugal forces to the capillary forces, namely the Centrifugal number, which together with the Stokes number characterizes the regime map of outcomes. This work provides the foundation for simulations of many-particle systems.

8:13AM A24.00002 Transverse alignment of fibers in a periodically sheared suspension: An absorbing phase transition with a slowly-varying control parameter , ALEXANDRE FRANCESCHINI, EMMANOUELA FILIPPIDI, Center for Soft Matter Research, Department of Physics, New York University, 10003 NY, USA, ELISABETH GUAZZELLI, IUSTI-CNRS UMR 6595, Polytech Marseille, Aix-Marseille Universite, 13453 Marseille, France, DAVID PINE, Center for Soft Matter Research, Department of Physics, New York University, 10003 NY, USA — Shearing fibers and polymer solutions tends to align particles with the flow direction. Here, we report that neutrally buoyant non-Brownian fibers subjected to oscillatory shear are observed to align perpendicular to the flow. This alignment occurs over a finite range of strain amplitudes and is governed by a subtle interplay between fiber orientation and short-range interactions through an athermal (non-equilibrium) process known as random organization. For a given strain amplitude and concentration, the mean field orientation defines a time-dependant control parameter that can drive the suspension through an absorbing phase transition. The slow drift of the control parameter does not influence the class of the transition. The measured critical threshold and exponents are consistent with the one reported for sphere suspensions. This work was supported by the NSF through the NYU MRSEC, Award DMR:0820341. Additional support was provided by a Lavoisier Fellowship (AF) and from the Onassis Foundation (EF).

8:26AM A24.00003 Acoustic metafluid with anisotropic mass density and tunable sound speed: An approach based on suspensions of orientable anisotropic particles , MARK SEITEL, STEPHEN TSE, JERRY SHAN, Rutgers University — We investigate liquid suspensions of micron-scale, anisotropic particles as potential acoustic metafluids having anisotropic and actively controllable acoustic properties. The effective mass density (and hence the sound propagation speed) of these metafluids can vary because the added mass of an anisotropic particle suspended in the fluid changes with the particle's orientation relative to the direction of the wave propagation. A suspension with disc-like particles oriented broadside to the direction of wave propagation is thus expected to have higher effective inertia and lower sound speed than a suspension with particles with end-on alignment. To test these predictions, sound speed is measured with a time-of-flight method in suspensions of micron-size nickel flakes suspended in oil, with and without magnetic-field-induced alignment of the particles. The sound speed, relative to the unaligned case, is found to decrease for particles oriented broadside to the sound wave, and increase for edgewise alignment. We also investigate the frequency dependence of the effective sound speed, since the added mass effect is expected to diminish as the flow becomes steady at low frequencies. The experimental results are compared to the predictions of a model proposed by Ahuja & Hardee (*J. Acoust. Soc. Am* 1978) for the acoustic properties of aligned oblate-spheroid suspensions.

8:39AM A24.00004 Secondary jets from high viscosity complex fluid[1] , GUSTAF MÅRTENSSON, THOMAS KURIAN, GUSTAV GRAEBER, Micronic Mydata AB — Jetting of high viscosity, particle filled fluids (e.g. solder paste) is of great interest in any number of industries, for example electronics manufacturing. A current problem with this technique is the presence of particle satellites outside the region of the main deposit caused partly by a secondary jet being emitted upon impact. Up to now most detailed studies of corona splashes have been performed mainly on Newtonian fluids.[2] We experimentally study the impact dynamics of a non-Newtonian fluid, lead-free solder paste, with a particle volume fraction of 0.45 and particle size range between 15 and 25 μm. The collected films show that over a range of impact velocities, secondary jets are only emitted after impact for high velocities. These observations are important in that they indicate an upper limit on the impact velocity that can be used in a real application. A threshold criteria for the production of secondary jets as a function of viscosity, η, surface tension, γ, and speed, v, will be presented.

[1]Funding from the Swedish Research Council (VR) is gratefully acknowledged.
[2]Xu, L. et al. 2005 "Drop Splashing on a Dry Smooth Surface," *PRL*, **94**, pp. 1-4

8:52AM A24.00005 Controlling Heap Formation in Microsphere Suspensions , CARLOS ORTIZ, KAREN DANIELS, ROBERT RIEHN, North Carolina State University — We explore how the formation of heaps of microspheres depends on the relative magnitudes of hydrodynamic, surface, and thermal forces. We assemble a heap of sub-micron polystyrene spheres by pumping a dilute ($\phi < 10^{-3}$) suspension towards a barrier in a microchannel. We map out a phase diagram to locate the heap-forming regime as a function of incoming bead velocity, controlled by driving pressure, and barrier surface charge, controlled by buffer conditions. As the barrier surface charge increases, the long-range repulsive normal force between the microspheres and the surface increases, causing the minimum driving pressure needed to form a pile to increase. We measure the static and dynamic properties of the heap as a function of system parameters.

9:05AM A24.00006 Concentrated suspensions in a homogeneous shear flow with finite fluid inertia , KYONGMIN YEO[1], MARTIN MAXEY, Brown University — The dynamics of monodisperse suspensions of neutrally buoyant particles under finite fluid inertia is investigated by numerical simulations using the force-coupling method. Here, we report on the effect of finite fluid inertia on the rheology and self-diffusion of the suspended particles for the volume fractions 0.2 - 0.4. It is found that the diffusivity is an increasing function of Reynolds number, while the particle-phase velocity fluctuations decrease at larger Reynolds number. Lagrangian velocity autocorrelation functions indicate that the motion of the suspended particles has a longer correlation under finite fluid inertia, which in turn contributes to the increase of the diffusivity. The changes in rheological parameters, such as the suspension viscosity and normal stress differences, under finite inertia are studied by analyzing pair-distribution functions. We found that the particle stresses become highly intermittent at larger Reynolds number.

[1]Now at Lawrence Berkeley National Laboratory

9:18AM A24.00007 Clouds of particles in a periodic shear flow[1] , BLOEN METZGER, IUSTI-CNRS, JASON E. BUTLER, University of Florida — We have investigated experimentally the time evolution of a cloud of non-Brownian particles submitted to a periodic shear flow in an otherwise pure liquid at low Reynolds number. This experiment is reminiscent to the famous Taylor experiment but applied to clouds of particles. It illustrates the irreversible nature of particulate systems submitted to a shear. Unlike Taylor's ink-drop, clouds of particles do not reconstitute when shear is reversed. We have found that, when repeating the cycles of shear, clouds of particles progressively disperse in the flow direction until they reach the threshold critical volume fraction. For very large initial volume fractions, clouds evolve into a "galaxy-like" shape.

[1]Visits and experiments were supported by the Partner University Fund on particulate flows and by ANR "ITS" JCJC SIMI 9

9:31AM A24.00008 Hydrodynamic effects on aggregation of colloidal particles[1] , JEFFREY MORRIS, Levich Institute, City College of New York — The effects of both hydrodynamic interaction and the form of the interparticle potential on the aggregation process for dispersed spherical particles are investigated by computational simulation. The simulation methods of Brownian Dynamics (BD) and Stokesian Dynamics (SD) are applied, over a range of solid volume fraction of $0.04 \leq \phi \leq 0.12$. The interparticle potential is a combination of a generalized Lennard-Jones form and a Yukawa potential, the latter of which describes a screened electrostatic repulsion at longer range. Hydrodynamic interactions were found to significantly reduce the solid fraction required for percolation, with the influence depending upon the form of the potential; the difference in percolation threshold was significant, with $\phi_{c,SD} \doteq 0.06$ and $\phi_{c,BD} \geq 0.08$ a typical difference for moderate repulsion barriers. These results are for O(1000) particles in a cubic unit cell.

[1]Supported by NSF PREM (DMR 0934206)

9:44AM A24.00009 A novel concept of dielectrophoretic engine oil filter[1] , BORIS KHUSID, YUEYANG SHEN, EZINWA ELELE , New Jersey Institute of Technology — A novel concept of an alternating current (AC) dielectrophoretic filter with a three-dimensional electrode array is presented. A filter is constructed by winding into layers around the core tube two sheets of woven metal wire-mesh with several sheets of woven insulating wire-mesh sandwiched in between. Contrary to conventional dielectrophoretic devices, the proposed design of electrodes generates a high-gradient field over a large working volume by applying several hundred volts at a standard frequency of 60 Hz. The operating principle of filtration is based on our recently developed method of AC dielectrophoretic gating for microfluidics. The filtration efficiency is expressed in terms of two non-dimensional parameters which describe the combined influence of the particle polarizability and size, the oil viscosity and flow rate, and the field gradient on the particle captivity. The proof-of-concept is tested by measuring the single-pass performance of two filters on positively polarized particles dispersed in engine oil: spherical glass beads, fused aluminum oxide powder, and silicon metal powder, all smaller than the mesh opening. The results obtained provide critical design guidelines for the development of a filter based on the retention capability of challenge particles.

[1]The work was supported in part by ONR and NSF.

9:57AM A24.00010 Making micro soccer balls: Spherical macro-clusters of colloidal particles by droplet evaporation on superhydrophobic surfaces , ALVARO MARIN, Physics of Fluids, University of Twente, ARTURO SUSARREY-ARCE, HAN GARDENIERS, Mesoscale Chemical Systems, University of Twente, DETLEF LOHSE, Physics of Fluids, University of Twente — Imagine a clean capillary droplet evaporating in a fakir state on a superhydrophobic micro-structured surface. When the superhydrophobicity is robust enough, the droplet will always remain on top of the micro-structure and the droplet will retain its spherical shape until its "death." Very often one can observe the remains of the impurities within the droplet left on top of the structure at the end of the process. In this work, instead of using clean liquid droplets, we use dilute colloidal dispersion droplets of monodisperse polymer micro-spheres (sizes from 0.2 to $2\mu m$). The colloidal dispersion droplet retains its spherical shape during its whole life, even when the whole solution has been evaporated. The remaining object consist on a spherical-shaped massive cluster of particles with diameters ranging from a few tens of microns up to several hundreds of microns, depending on the amount of micro-particles present in the solution and on the final packing fraction. We will discuss on the different observed packing fractions, particle arrangements and their governing parameters. Additionally some predictions will also be introduced.

Sunday, November 20, 2011 8:00AM - 10:10AM – Session A27 Focus Session: Wind Energy Fluid Dynamics I Ballroom I-IV

8:00AM A27.00001 Computational and Experimental Studies of Turbulence in Wind and Hydrokinetic Energy: From Turbines to Farms[1] , FOTIS SOTIROPOULOS, SEOKKOO KANG, XIAOLEI YANG, LEONARDO CHAMORRO, CRAIG HILL, ROGER ARNDT, St. Anthony Falls Laboratory, University of Minnesota — Recent computational and experimental advances at the St. Anthony Falls Laboratory (SAFL) aimed at understanding the structure of turbulence past wind and hydrokinetic turbines and farms will be presented. A powerful computational framework has been developed for carrying out LES of turbulent flow past complete turbine configurations as well as large-scale wind farms. For the former, the geometrical details of the turbine are resolved on fine computational grids using the CURVIB method with a wall model (Kang et al., Adv. in Water Resources, 34(1), 98-113, 2011) while for the latter the turbines are parametrized as actuator disks. Laboratory experiments in the SAFL atmospheric boundary layer wind tunnel and a large water flume have provided data sets for model validation. The computed and experimental results yield novel insights into the structure of turbulence in turbine wakes and suggest strategies for optimizing layouts of multi-turbine arrays for maximizing energy capture.

[1]This work was supported by Department of Energy DOE (DE-EE0002980), Xcel Energy through the Renewable Development Fund (grant RD3-42), Verdant Power Inc., and the University of Minnesota Initiative for Renewable Energy and the Environment.

8:13AM A27.00002 Panel method for the wake effects on the aerodynamics of vertical-axis wind turbines , UDIT GOYAL, Graduate Student, DIETMAR REMPFER, Associate Professor — A formulation based on the panel method is implemented for studying the unsteady aerodynamics of straight-bladed vertical-axis wind turbines. A combination of source and vortex distributions is used to represent an airfoil in Darrieus type motion. Our approach represents a low-cost computational technique that takes into account the dynamic changes in angle of attack of the blade during a cycle. A time-stepping mechanism is introduced for the wake convection, and its effects on the aerodynamic forces on the blade are discussed. The focus of the study is to describe the effect of the trailing wakes on the upstream flow conditions and coefficient of performance of the turbines. Results show a decrease in Cp until the wake structure develops and assumes a quasi-steady behavior. A comparison with other models such as single and multiple streamtubes is discussed, and optimization of the blade pitch angle is performed to increase the instantaneous torque and hence the power output from the turbine.

8:26AM A27.00003 Fluid-Structure Interaction Modeling of Horizontal- and Vertical-Axis Wind Turbines , YURI BAZILEVS — In this talk I will present a collection of numerical methods combined into a single framework for wind turbine modeling and simulation. I will cover our turbulence modeling and discretization approach, structural modeling and discretizations for wind turbine blades, the details of fluid-structure interaction (FSI) computational procedures focusing on the challenges of FSI coupling, and fluid mechanics domain mesh motion strategies in the presence of large rotation. I will present simulations of the NREL 5MW offshore baseline wind turbine rotor, including validation against published data. I will conclude by showing preliminary simulations of a vertical-axis wind turbine design.

8:39AM A27.00004 Quantitative Full-Scale Wind Turbine Flow Measurements[1] , MATTHIAS KINZEL, QUINN MULLIGAN, JOHN DABIRI, Caltech — To analyze the interaction between vertical axis wind turbines (VAWT) in detail it is important to gain a deeper understanding of their flow field. Quantitative in situ measurements pose a great challenge because of the large spatial dimensions, high flow velocities and remote locations of the VAWT. The aim of this work is to perform Particle Image Velocitmetry (PIV) in a horizontal cross section of a VAWT. The major difficulty is the choice of adequate seeding particles and illumination method for the large field of view, which is necessary. The flow velocities on the other hand require a high speed camera and the whole setup has to be powered self-sufficiently. However, PIV yields a two dimensional two component velocity field together with the out of plane component of vorticity and is therefore a considerable advantage over the single point measurements which are available today. The presentation will deal with different methods for seeding the flow. The properties of these flow tracers will be discussed and their ability to follow the flow reliably evaluated. Preliminary PIV results of the wind velocities at the test site will be shown.

[1]The financial support of the Moore Foundation is gratefully acknowledged.

8:52AM A27.00005 Novel Diagnostics to Investigate Turbine-Turbine Interactions[1] , B.J. BALAKUMAR, S. POL, Los Alamos National Laboratory — Strong aerodynamic interactions between upstream wind turbine wakes and downstream turbine blades cause fatigue loads and reduce turbine reliability. The wake structure also mediates the vertical flux of momentum and affects the power output of downstream turbines in turbine arrays. Despite their importance, our current understanding of wake-turbine interactions and wake structure is limited, especially under complex operating conditions such as yawed inflow and dynamic stall. Traditional diagnostics such as sonic anemometers, hotwires, and lidars suffer from interference and accuracy limitations and prove inadequate. At LANL, we have developed a Large Field-of-View Particle Image Velocimeter (LF-PIV) capable of measuring 3 m$\times$1 m (per camera) wake and inflow regions around a 5m-scale turbine. This scalable diagnostic operates in conjunction with a hub-mounted Rotating PIV (R-PIV) diagnostic to observe blade boundary layer and separation while the turbine is in operation. We discuss the diagnostic development challenges, solutions used to overcome these, and the interesting physics that these diagnostics promise to illuminate.

[1]Supported by LANL LDRD grant 20100040DR (PI: Curtt Ammerman).

9:05AM A27.00006 Large Field of View Particle-Image Velocimetry (LF-PIV): Design and Performance[1] , SUHAS POL, JOHN HOFFMAN, BALAKUMAR BALASUBRAMANIAM, P-23, LANL TEAM — We discuss the challenges and limitations associated with the development of a Large Field of View Particle Image Velocimetry (LF-PIV) diagnostic that is capable of resolving large scale motions (3m x 1m per camera) in gas phase laboratory experiments. While this diagnostic is developed for the measurement of wakes and local inflow conditions around research wind turbines, the design considerations provided here are also relevant for the application of LF-PIV to atmospheric boundary layer, rotorcraft dynamics and large-scale wind tunnel flows. Scaling laws associated with LF-PIV systems are presented along with the performance analysis of low-density, large diameter Expancel particles that appear to be promising candidates for LF-PIV seeding. Comparison of data obtained by LF-PIV measurements (2MP camera) and regular format sized PIV measurements show an agreement of within 1% for mean velocity and 8% for turbulent statistics respectively.

[1]Los Alamos National Laboratory, NM, USA

9:18AM A27.00007 Investigation on wind turbine wakes: wind tunnel tests and field experiments with LIDARs , GIACOMO VALERIO IUNGO, EPFL, TING WU, JULIETTE CÖEFFÉ, FERNANDO PORTÉ-AGEL, WIRE TEAM — An investigation on the interaction between atmospheric boundary layer flow and wind turbines is carried out with wind tunnel and LIDAR measurements. The former were carried out using hot-wire anemometry and multi-hole pressure probes in the wake of a three-bladed miniature wind turbine. The wind turbine wake is characterized by a strong velocity defect in the proximity of the rotor, and its recovery is found to depend on the characteristics of the incoming atmospheric boundary layer (mean velocity and turbulence intensity profiles). Field experiments were performed using three wind LIDARs. Bi-dimensional scans are performed in order to analyse the wake wind field with different atmospheric boundary layer conditions. Furthermore, simultaneous measurements with two or three LIDARs allow the reconstruction of multi-component velocity fields. Both LIDAR and wind tunnel measurements highlight an increased turbulence level at the wake boundary for heights comparable to the top-tip of the blades; this flow feature can produce dangerous fatigue loads on following wind turbines.

9:31AM A27.00008 Experimental study of the kinetic energy budget of a wind turbine stream-tube[1] , JOSE LEBRON, Rensselaer Polytechnic Institute, LUCIANO CASTILLO, Texas Tech University, CHARLES MENEVEAU, Mechanical Engineering and CEAFM, Johns Hopkins University — We consider the kinetic energy budget of a wind turbine stream-tube. Unlike the traditional Betz analysis, which neglects turbulence, we include the effects of turbulent kinetic energy fluxes at the stream-tube boundaries as well as dissipation of kinetic energy inside the stream-tube. The analysis is applied to a previously acquired PIV data set described in Cal et al. (2010, JRSE 2, 013106). Since the wind turbines in that experiment were lightly loaded, we observe that the stream-tube diameter changes by only 7% and the mean velocity by 12% between front and back of the wind turbine. Deviations from axisymmetry of the stream-tube are observed downstream of the wind turbine, mainly due to tower effects. Besides the importance of mean kinetic energy flux, we find that the radial Reynolds stress component acting on the stream-tube surface is a dominant contributor to the overall kinetic energy balance, and is responsible for the wake recovery. We also attempt to evaluate the turbulent dissipation rate by integrating the Reynolds stress times the mean velocity gradients inside the stream tube.

[1]Support by NSF GK-12 Program No. 0742436 and NSF CBET 0730922

9:44AM A27.00009 Interaction of an Artificially Thickened Boundary Layer with a Vertically Mounted Pitching Airfoil[1] , TRISTEN HOHMAN, ALEXANDER SMITS, LUIGI MARTINELLI, Princeton University — Wind energy represents a large portion of the growing market in alternative energy technologies and the current landscape has been dominated by the more prevalent horizontal axis wind turbine. However, there are several advantages to the vertical axis wind turbine (VAWT) or Darrieus type design and yet there is much to be understood about how the atmospheric boundary layer (ABL) affects their performance. In this study the ABL was simulated in a wind tunnel through the use of elliptical shaped vortex generators, a castellated wall, and floor roughness elements as described in the method of Counihan (1967) and then verified its validity by hot wire measurement of the mean velocity profile as well as the turbulence intensity. The motion of an blade element around a vertical axis is approximated through the use of a pitching airfoil. The wake of the airfoil is investigated through hot wire anemometry in both uniform flow and in the simulated boundary layer both at $Re = 1.37 \times 10^5$ based on the chord of the airfoil.

[1]Sponsored by Hopewell Wind Power (Hong Kong) Limited

9:57AM A27.00010 Effect of Free-stream Turbulence on Flow Separation in S809 Wind Turbine Blade[1] , SHEILLA TORRES-NIEVES, VICTOR MALDONADO, Rensselaer Polytechnic Institute, CHARLES MENEVEAU, The Johns Hopkins University, LUCIANO CASTILLO, Texas Tech University — Two-dimensional Particle Image Velocimetry (2D-PIV) measurements are performed to study the effects of free-stream turbulence on the flow around a smooth and rough surface airfoil, specifically under stall conditions. A 0.25-m chord model with an S809 profile, common for horizontal-axis wind turbine applications, was tested at a wind tunnel speed of 10 m/s, resulting in Reynolds numbers based on the chord of 182,000 and turbulence intensity levels of up to 6.14%. Analysis of the mean flow over the suction surface shows that, contrary to what is expected, free-stream turbulence is actually advancing separation, particularly when the turbulent scales in the free-stream are of the same order as the chord. This behavior is also confirmed by the examination of the aerodynamic coefficients; under stall conditions, the aerodynamic performance is slightly improved, from L/D 1.696 to 1.787. Further analysis of mean and turbulent quantities in the flow field will be performed in order to understand the mechanism by which free-stream turbulence is advancing separation.

[1]Research support: ONR, NSF (Graduate Research Fellowship to S.T.N., grants CBET-0553314, 0730922, 1033942, and HRD-0202171) and Ford Foundation Predoctoral Diversity Fellowship to S.T.N.

Sunday, November 20, 2011 10:15AM - 12:10PM –

Session B27 Awards Presentations, Followed by the Otto Laporte and Corrsin Lectures Ballroom I-IV

10:15AM B27.00001 Welcome, Presentation of Awards and DFD Fellowships –

10:40AM B27.00002 Otto Laport Lecture: A Fascination with Fluids: Vortices and Vortex Breakdown. , TONY MAXWORTHY, University of Southern California — The problem of the dynamics of long slender vortices, e.g., tornadoes, dust devils, waterspouts, fire whirls, internal flow in rotating machinery, leading edge and trailing vortices on lifting surfaces, etc, has been a fascination for me for close to 50 years. A sequence of experimental studies will be presented, together with reasoned physical explanations and related theoretical arguments, that attempt to bring some order to the sometimes-controversial discussion that has swirled about the subject during that time.

11:25AM B27.00003 Stanley Corrsin Award Talk: Big and small swirls in the maze: Modeling turbulence in Large Eddy Simulations , CHARLES MENEVEAU, Johns Hopkins University — The existence of strongly coupled motions at many length and time scales is one of the most challenging aspects of turbulent flows. We describe this challenge in the specific context of Large Eddy Simulations and subgrid-scale modeling. Some examples of progress achieved through scale-aware self-consistency conditions, of which the dynamic model is a well-known example, will be presented. Laboratory and field data that have been used to illuminate various aspects of the problem will also be discussed.

Sunday, November 20, 2011 1:30PM - 2:05PM –

Session C27 Invited Session: Foam flow and liquid films motion: role of the surfactants properties Ballroom I-II

1:30PM C27.00001 Foam flow and liquid films motion: role of the surfactants properties[1] , ISABELLE CANTAT, Institut de Physique de Rennes — Liquid foams absorb energy in a much more efficient way than each of its constituents, taken separately. However, the local process at the origin of the energy dissipation is not entirely elucidated yet, and several models may apply, thus making worth local studies on simpler systems. We investigate the motion through a wet tube of transverse soap films, or lamellae, combining local thickness and velocity measurements in the wetting film. For foaming solution with a high dilatational surface modulus, we reveal a zone of several centimeters in length, the dynamic wetting film, which is significantly influenced by a moving lamella. The dependence of this influence length on lamella velocity and wetting film thickness provides an accurate discrimination among several possible surfactants models.

[1]In collaboration with B. Dollet

Sunday, November 20, 2011 1:30PM - 2:05PM –

Session C29 Invited Session: Biomimetics and Tubercles on Flippers for Hydrodynamic Flow Control Ballroom III-IV

1:30PM C29.00001 Biomimetics and Tubercles on Flippers for Hydrodynamic Flow Control, FRANK E. FISH, Department of Biology, West Chester University — The biomimetic approach seeks to incorporate designs based on biological organisms into engineered technologies. Biomimetics can be used to engineer machines that emulate the performance of organisms, particularly in instances where the organism's performance exceeds current mechanical technology or provides new directions to solve existing problems. The ability to control the flow of water around the body dictates the performance of marine mammals in the aquatic environment. Morphological specializations of marine mammals afford mechanisms for passive flow control. Aside from the design of the body, which minimizes drag, the morphology of the appendages provide hydrodynamic advantages with respect to drag, lift, thrust, and stall. Of particular interest are the pectoral flippers of the humpback whale (*Megaptera novaeangliae*). These flippers act as wing-like structures to provide hydrodynamic lift for maneuvering. The use of any such wing-like structure in making small radius turns to enhance both agility and maneuverability is constrained by performance associated with stall. Delay of stall can be accomplished passively by modification of the flipper leading edge. The design of the flippers includes prominent leading edge bumps or tubercles. Such a design is exhibited by the leading edge tubercles on the flippers of humpback whales. These novel morphological structures induce a spanwise flow field of separated vortices alternating with regions of accelerated flow. The coupled flow regions maintain areas of attached flow and delay stall to high angles of attack. The morphological features of humpback whales for flow control can be utilized in the biomimetic design of engineered structures and commercial products for increased hydrodynamic performance. Nature retains a store of untouched knowledge, which would be beneficial in advancing technology.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D1 Geophysical Flows: Atmospheric II 301

2:10PM D1.00001 Thermally driven upslope flow in mountainous terrain, DAN LIBERZON, CHRISTOPHER HOCUT, HARINDRA J. FERNANDO, Civil Engineering and Geological Sciences University of Notre Dame, Notre Dame, IN 46556, ENVIRONMENTAL FLUID DYNAMICS TEAM — Buoyancy driven up-slope flow and its separation from mountain apex are two important processes that determine meso and regional flows in mountainous areas. Such flow configurations have applications from mountain meteorology to large scale monsoonal circulation. A combined experimental and theoretical study toward improving our understanding of the mechanisms governing upslope flow processes, in particular, generation of upstream circulating cells and plume rise at the apex is presented. The experiments were performed in a 1.25x.35x.3 m water tank, using an inclined (10 to 30 degrees from the horizontal) electrical foil as the heated slope. Under certain condition the flow configuration produced stable circulation cells and rising limited plumes of finite height. Particle Tracking Velocimetry and flow visualization techniques were used for the diagnostics of velocity field and plume rise height, and relevant salient dimensionless quantities were evaluated in terms of governing parameters. Theoretical arguments are presented to explain the results. Parameter ranges for the appearance of characteristic flow patterns are also delineated.

2:23PM D1.00002 Thermal stability effects on the turbulent boundary-layer flow over a steep 2-D hill: Near-wake structure, WEI ZHANG, University of Minnesota, FERNANDO PORTE-AGEL, EPFL — Thermally stratified boundary-layer flows over complex topography are of great interest to the atmospheric science and wind engineering communities. Neutral boundary-layer flows over topography have been extensively studied by wind-tunnel experiments and numerical simulation techniques such as Large-Eddy Simulation (LES). Thermal stability effect, however, is seldom considered due to difficulty of physical simulation in wind tunnels. Experimental studies of thermal stability (neutral, stable and convective) effects on the flow over a steep 2-D hill were conducted in an atmospheric boundary-layer wind tunnel. The 2-D model hill had a steepest slope of 0.73 and its shape followed a cosine shape with height of 7 cm and length of 14.5 cm. High-resolution PIV was employed to characterize the onset of separation, the recirculation zone and flow reattachment location. Results show that thermal stability significantly affects the turbulent flow downwind of the hill, with an elongated recirculation zone and delay of the reattachment in the stable case, and a shorter recirculation zone and early attachment in the convective case. The present study leads to better understanding of thermal stability effects on the boundary-layer flow over a steep 2-D hill, and provides a reliable database for validation and improvement of LES models.

2:36PM D1.00003 Atmospheric stability analysis over statically and dynamically rough surfaces[1], EMINA MARIC, MEREDITH METZGER, University of Utah, ARINDAM SINGHA, REZA SADR, Texas A&M University at Qatar — The ratio of buoyancy flux to turbulent kinetic energy production in the atmospheric surface layer is investigated experimentally for air flow over two types of surfaces characterized by static and dynamic roughness. In this study, "static" refers to the time-invariant nature of naturally-occurring roughness over a mud/salt playa; while, "dynamic" refers to the behavior of water waves along an air-water interface. In both cases, time-resolved measurements of the momentum and heat fluxes were acquired from synchronized 3D sonic anemometers mounted on a vertical tower. Field campaigns were conducted at two sites, representing the "statically" and "dynamically" rough surfaces, respectively: (1) the SLTEST facility in Utah's western desert, and (2) the new Doha airport in Qatar under construction along the coast of the Persian Gulf. Note, at site 2, anemometers were located directly above the water by extension from a tower secured to the end of a 1 km-long pier. Comparisons of the Monin-Obukhov length, flux Richardson number, and gradient Richardson number are presented, and discussed in the context of the observed evolution of the turbulent spectra in response to diurnal variations of atmospheric stability.

[1]Supported by the Qatar National Research Fund.

2:49PM D1.00004 LES of atmospheric boundary layer flow over fluvial-like anisotropic topography with a dynamic surface drag model[1], WILLIAM ANDERSON, Mechanical Eng, Baylor University, CHARLES MENEVEAU, Mechanical Eng. & CEAFM, Johns Hopkins University — A dynamic surface drag model (A. & M. 2011, JFM 679, 288 - 314) is applied in LES of atmospheric boundary layer (ABL) flow over fractal-like topography where the height field exhibits power-law energy spectrum. Initially, the dynamic drag model was applied in LES of ABL flow over isotropic synthetic fractal-like roughness. Here we consider fluvial-like anisotropic landscapes. Two main cases are considered. The first is a fluvial-like topography built through numerical solution of the Kardar-Parisi-Zhang equation.[2] The second is a rescaled topography (Texas) map from the U.S. National Elevation Dataset. These landscapes are dominated by anisotropic modes that have emerged through geomorphological erosion processes. The dynamic model yields stable solutions even in these highly anisotropic cases: performance is strongest for cases where the LES grid- and test-filter width are within the landscape "self-similar" range. Weaknesses are reported for cases where spectral exponent changes with wavenumber, motivating the development of a scale-dependent version of the dynamic approach using two test-filters.

[1]Supported by NSF (AGS-1045189).
[2]Thanks also to Profs. P. Passalacqua and F. Porte-Agel for providing KPZ solution fields.

3:02PM D1.00005 Lagrangian coherent structures in hurricanes, DOUG LIPINSKI, KAMRAN MOHSENI, University of Florida — We present the results of a "surface tracking" algorithm for efficiently computing Lagrangian coherent structure (LCS) surfaces in three dimensions. The algorithm is applied to data from a Weather Research and Forecasting simulation of hurricane Rita. The highly complicated LCS surfaces reveal complex dynamics and transport in the hurricane, particularly in the lower atmosphere boundary layer and the upper level outflow. The lower level transport in the hurricane is of particular importance for accurate intensity prediction in hurricane forecasts due to the uncertainty in the ocean-atmosphere interaction. Understanding the lower level transport and mixing behavior in hurricanes could lead to significant advances in hurricane intensity prediction.

3:15PM D1.00006 Description of coherent features in the atmospheric boundary layer by low-dimensional analysis of surface pressures[1]

, GREGORY LYONS, NATHAN MURRAY, University of Mississippi - NCPA — Short-term localized forecasting of the near-earth atmospheric boundary layer could enhance empirical models and provide input for wind energy control systems. It is expected that a successful forecasting method could be developed through characterization of the coherent turbulent structures in the near-earth flow field. The coherence of these features suggests description by a low-dimensional method, such as the dynamic mode decomposition. To this end, a circular array of pressure transducers approximately 60 meters in diameter was deployed on the earth's surface, and time series were recorded at each sensor location. As scalar observables, these pressure measurements are functions of the flow field system. As such, by decomposing these data into dynamic modes, the spatial and temporal coherence of local features in the near-earth atmospheric boundary layer is described.

[1]Supported by the U.S. Department of Energy.

3:28PM D1.00007 A comparison of scaling in satellite and in situ water vapor measurements

, KYLE PRESSEL, WILLIAM COLLINS, The University of California, Berkeley, ANKUR DESAI, The University of Wisconsin, Madison — Improvements in the characterization of cloud scale variability of water vapor are a direct means of improving stochastic parametrization of clouds in global climate models (GCMs). The Atmospheric Infrared Sounder (AIRS) provides nearly global physical retrievals of the vertical profile of water vapor mass mixing ratio, at resolutions far greater than cloud scale. The 447m WLEF tower located near Park Falls, WI is instrumented to provide high frequency (10Hz) observations of water vapor at 396, 122, and 30m above ground level. Using Taylor's hypothesis, the high frequency tower observations can be used to characterize the variability of water vapor at or near cloud scales, but only at a single location. In this work, we will compare the climatology of water vapor scaling (scale-invariance) observed by AIRS and from the WLEF tower, in an effort to examine the potential use of the scale-invariant properties of the water vapor field to gain global information about cloud scale water vapor variability from AIRS.

3:41PM D1.00008 Investigating multiscale modeling with the Kuramoto-Sivashinsky equation

, LAUREN PADILLA, CLARENCE ROWLEY, GEOFFREY VALLIS, Princeton University — The multiscale modeling framework (MMF), also known as superparameterization, is a new approach to solving the sub-gridscale closure problem in weather and climate simulations that are too expensive for direct numerical simulation. In the MMF, the true equations of motion for the sub-gridscale processes replace typical parameterizations which provide closure based on large-scale variables alone where no local dynamics are allowed to develop. The extent to which MMF represents improvement over conventional models is an open question that we investigate systematically in a simple fluid-like system, the Kuramoto-Sivashinsky (K-S) equation. This one-dimensional, non-linear, partial differential equation is an ideal test case for the MMF approach. The K-S equation exhibits chaotic behavior and includes the effects of production, dissipation, and advection yet is simple enough to sample many initial conditions over long time periods. We present results comparing the energy spectra, error, and turbulent kinetic energy of solutions achieved through direct numerical simulation, MMF with different coupling schemes, and conventional large-scale closure.

3:54PM D1.00009 A leaky rigid lid[1]

, LYUBOV CHUMAKOVA, RUBEN ROSALES, MIT, ESTEBAN TABAK, Courant, NYU — Various models of the atmosphere use the rigid lid approximation. In reality the atmosphere does not have a definite top. Furthermore, mathematically the two problems are drastically different. In the study of linear problem in hydrostatic balance the rigid lid approximation leads to a problem with discrete spectrum, while a semi-infinite atmosphere problem has a continuous one. The energy that is projected onto the continuous part of the spectrum is disregarded in the rigid lid approach. We propose to fix this "leak" problem via changing the boundary conditions at the top of the rigid lid.

[1]Supported by NSF DMS-0903008.

4:07PM D1.00010 Parameterization of turbulence characteristics of Atmospheric surface layer in Qatar

, ARINDAM SINGHA, Postdoctoral Research Associate, REZA SADR, Assistant Professor — Turbulent characteristics of atmospheric boundary layer are of utmost importance in modeling the large-scale meteorological processes, diffusion of atmospheric contaminants, heat transfer and evaporation from the earth surface. Available data are for some areas on the globe and are really sparse in tropical regions, except a few recent studies in Asia. There had been some recent studies in tropical weather in southwestern Asia but no study is carried out in Persian Gulf region. An atmospheric measurement station has been designed and installed in a site in the coastal region of Doha, Qatar, to characterize the nature of atmosphere surface layer (ASL) and ocean wave characteristics in this field. The aim of the present study is to report the micrometeorological data collected from this site. The normalized variation of the turbulent velocity components and temperature were studied using Monin-Obukhov similarity theory (MOST). This study also attempts to verify the validity of MOST in the context of the data collected for a marine ASL in Qatar, and compares the modeling parameters with other investigations around the world. This is the first ever study of ASL in this area, and is expected to be a foundation of further atmospheric research endeavors in Qatar.

Sunday, November 20, 2011 2:10PM - 4:20PM –

Session D2 Nanofluids I 302

2:10PM D2.00001 Investigation of Flow Boundary Conditions and Diffusion in Nanochannels using Molecular Dynamics Simulations

, ALI KHARAZMI, NIKOLAI PRIEZJEV, Michigan State University — The influence of nano-confinement and slip boundary conditions on the thermal motion of fluid molecules is investigated using the LAMMPS molecular dynamics program. We consider a neutral Lennard-Jones fluid confined by crystalline walls with weak surface energy. Particular attention was paid to the implementation of a wall model that allows fine adjustment of the slip length without changing the interfacial fluid structure. We used the original Steele's surface energy decomposition, i.e., that the interaction potential between a fluid molecule and a solid substrate can be expressed as a Fourier expansion in the reciprocal-lattice vectors of the substrate surface. The local diffusion coefficients in the directions parallel and perpendicular to confining walls were estimated as a function of a distance from the walls and then correlated with the slip length. The results of this numerical study are important for interpretation of nano-PIV (Particle Image Velocimetry) measurements of interfacial shear flows and boundary slippage. Financial support from the National Science Foundation (CBET-1033662) is gratefully acknowledged.

2:23PM D2.00002 Studying nanoscale slip mechanisms of simple fluids at low and high shearing velocities by molecular dynamics

XIN YONG, LUCY ZHANG, Rensselaer Polytechnic Institute — Understanding slip is of great importance, especially in the emerging research topics and applications of nanofluidics. In this study, we reveal the nanoscale slip mechanisms in different flow regimes that are characterized by shearing velocities. They are modeled using direct molecular dynamics simulation of a nanoscale Couette flow system. An isothermal fluid system is built by the Dissipative Particle Dynamics thermostat, with either rigid or thermal walls. The purpose of using isothermal simulation is to simplify the discussion by excluding the significant influence on the slip from dramatic viscous heating at high shear rates. Our results show an abrupt jump of slip length when the shearing velocity is increased to a critical value, implying a transition in the slip mechanisms. The transition is followed by a linear increase in slip length, with a slope that is dependent on the characteristic of the walls and wall models. Increasing the rigidity of the thermal walls results in slip behaviors that asymptotically approach the one of a rigid wall. The slip behavior supports the two mechanism hypothesis shown in Martini et al.,[1] but with a different interpretation for the slip mechanism at high shearing velocities.

[1]A. Martini, H.-Y. Hsu, N. A. Patankar, and S. Lichter, Phys. Rev. Lett. 100, 206001 (2008).

2:36PM D2.00003 A High-Order Accurate Numerical Solution of Incompressible Microchannel Flows in Slip and Transition Flow Regimes[1]

KAZEM HEJRANFAR, MIR HAMED MOHAFEZ, ALI KHAJEH-SAEED, Aerospace Engineering Department, Sharif University of Technology, Tehran, Iran — In this study, a high-order accurate numerical solution of steady incompressible flows in 2D microchannels is presented. The numerical method used is a fourth-order compact implicit operator scheme which is efficiently implemented to solve the incompressible Navier-Stokes equations in the primitive variables formulation using the artificial compressibility method. The present methodology considers the solution of the Navier-Stokes equations with employing different slip boundary conditions on the wall to model the slip flow in microchannels. Since the slip boundary conditions contain the derivatives of the u velocity, using the compact method the slip boundary conditions can be easily and accurately implemented. Herein, different slip boundary conditions are used and the results for fully developed velocity profiles in the slip flow regime ($Kn < 0.1$) and in the transition flow regime ($Kn > 0.1$) are compared with the available numerical and analytical results which show good agreement.

[1]The authors would like to thank Sharif University of Technology for financial support of this research.

2:49PM D2.00004 Gas Flow near a Smooth Plate

WILLIAM A. DUCKER, ADAM P. BOWLES — We examine gas flow adjacent to a molecularly smooth solid, muscovite mica. The fluctuations in force acting on a glass sphere as a function of proximity to a mica plate were measured in air, and were used to obtain the damping. The damping was interpreted as a lubrication force. The measured damping as a function of separation in the slip-flow regime corresponds to a slip length of 480 $\pm$ 70 nm, which is equivalent to highly specular gas molecule collisions. The slip flow model also fits the data for separations as small as one mean free path.

3:02PM D2.00005 Molecular diffusion and tensorial slip at surfaces with periodic and random nanoscale textures

NIKOLAI PRIEZJEV, Michigan State University — The influence of periodic and random surface textures on the flow structure and effective slip length in Newtonian fluids is investigated by molecular dynamics (MD) simulations. This study is motivated by the possibility to generate transverse flows in microfluidics devices to enhance mixing and separation processes. We consider a situation where the typical pattern size is smaller than the channel height and the local boundary conditions at wetting and nonwetting regions are characterized by finite slip lengths. In case of anisotropic textures, the interfacial diffusion coefficient of fluid molecules near heterogeneous surfaces correlates well with the effective slip length as a function of the shear flow direction with respect to the texture orientation. In addition, it was found that the angular dependence of the effective slip length obtained from MD simulations is in good agreement with hydrodynamic predictions provided that the pattern size is larger than several molecular diameters. These findings lend support for the microscopic justification of recently introduced tensor formulation of the effective slip boundary conditions in the case of noninertial flows of Newtonian fluids over smooth surfaces with nanoscale anisotropic textures. Funding from NSF (CBET-1033662) is gratefully acknowledged.

3:15PM D2.00006 Investigation of Gas and Fluid Flow through a Single Nanopore

SERAH FRIEDMAN, ANGEL VELASCO, ZUZANNA SIWY, PETER TABOREK, University of California, Irvine — Mass flow through ion-track etched nanopores with diameters ranging in size from 100 to 300 nm was measured using mass spectrometry. The thickness of the membrane was 12 micron, so our system was modeled after flow through a long pipe. The mass flow was caused by a pressure difference across the membrane of order 1 atm, with the low pressure side at vacuum. At room temperature diffusive transport through the membrane was comparable to mass flow through the hole so the temperature was lowered to -50 C which stopped the diffusive transport. The flow rates for gaseous Helium, Argon, and Nitrogen were studied and agreed with theory at a variety of Knudsen numbers. In contrast, with liquid water or Nitrogen on the high pressure side of the membrane, preliminary results show higher flow rates than can be accounted for by classical viscous flow theory.

3:28PM D2.00007 Surface-Gas Interaction Effects on Nanoscale Flows

MURAT BARISIK, ALI BESKOK, Old Dominion University — Three-dimensional molecular dynamics (MD) simulations of linear Couette flow of argon gas confined within nano-scale channels are investigated as a function of the surface-gas interaction strength ratio (ISR). Simulations are performed in the slip, transition and free molecular flow regimes by keeping the base pressure constant for different ISR cases. Near-wall gas density increases with increased ISR and eventually results in adsorption of argon on the surfaces. Although the velocity profiles agree with the kinetic theory predictions in the bulk of the channel, they show sudden increase in the near wall region, resulting in decreased velocity slip at the interface. High ISR values are shown to induce velocity stick. Increase in the ISR results in stronger surface–particle interactions. Hence, the surface virial becomes more dominant in the near wall region, resulting in increasingly anisotropic normal stresses. Utilizing the kinetic theory and MD predicted shear stress values, the tangential momentum accommodation coefficient for argon gas, interacting with FCC structured walls (100) plane facing the fluid, is shown to increase with increased ISR.

3:41PM D2.00008 Confinement Effects in Oscillating Nanoscale Flows

CHARLES LISSANDRELLO, VICTOR YAKHOT, KAMIL L. EKINCI, Department of Mechanical Engineering, Boston University — We present an experimental study of confinement in oscillating nanoscale flows. In the experiment, we measure the resonant parameters of a microcantilever with a sphere attached at the tip. As the sphere is brought towards a flat plate in a dry nitrogen environment, the flow becomes confined in the gap between the sphere and the plate, affecting the resonant frequency and dissipation of the cantilever. We tune the gap, the nitrogen pressure, and the oscillation frequency in order to study the flow over a broad range of dimensionless parameters. We observe deviations from continuum fluid dynamics at small gaps, low pressures, and high frequencies. Using these measurements, we provide an in-depth characterization of confinement effects in oscillating nanoflows.

3:54PM D2.00009 Small Scale Effect on Flow-Induced Instability of Double-Walled Carbon Nanotubes
, TAI-PING CHANG, National Kaohsiung First University of Science & Technology, MEI-FENG LIU, I-Shou University — Small scale effect on flow-induced instability of double-walled carbon nanotubes (DWCNTs) is investigated using an elastic shell model based on Donnell's shell theory. The dynamic governing equations of DWCNTs are derived on the basis of nonlocal elasticity theory, in addition, the van der Waals (vdW) interaction between the inner and outer walls is considered in the shell modeling. The instability of DWCNTs that is induced by a pressure-driven steady flow is studied. The numerical computations indicate that as the flow velocity increases, DWNCTs have a way to get through multi-bifurcations of the first and second bifurcations in turn. It can be concluded that the critical flow velocity is closely correlated to the ratio of the length to the radius of DWCNTs, the pressure of the fluid and the small scale effects. Furthermore, it is interesting to notice that as the small scale effects are considered, the first critical flow velocities of DWCNTs decrease as compared to the results with the classical (local) continuum mechanics, therefore, the small scale effects play an important role on performing the instability analysis in the fluid-conveying DWCNTs.

4:07PM D2.00010 Fast Water Transport in CNTs: length dependence and entrane/exit effects
, JENS HONORE WALTHER, TU Denmark, PETROS KOUMOUTSAKOS, ETH Zurich — Superfast water transport in carbon nanotube (CNT) membranes has been reported in experimental studies. We use Molecular Dynamics simulations to elucidate the mechanisms of water entry, exit and transport in $2\,nm$-diameter hydrophobic CNTs embedded in a hydrophilic membrane matrix. We demonstrate, for the first time, that under imposed pressures of the order of 1 bar, water entry into the CNT cavity and exit from the CNT end, can occur only on pre-wetted membranes. We conduct large scale simulations for up to $500\,nm$ long CNTs and observe a previously unseen dependence of the flow enhancement rates on the CNT length. We relate the present findings to past computational and experimental studies, we discuss previous continuum assessments for this flow and propose underlying physical mechanisms.

Sunday, November 20, 2011 2:10PM - 4:07PM –
Session D3 Multiphase Flow II: Cavitation 303

2:10PM D3.00001 Ventilated Supercavities[1]
, ELLISON KAWAKAMI, University of Minnesota, ROGER ARNDT — Drag reduction and/or speed augmentation of marine vehicles by means of supercavitation is a topic of great interest. During the initial launch of a supercavitating vehicle, ventilation is required to supply an artificial cavity until conditions at which a natural supercavity can be sustained are reached. A study has been carried out to determine various aspects of the flow physics of a supercavitating vehicle at Saint Anthony Falls Laboratory. During the present experimental work, the ventilated supercavity formed behind a sharp-edged disk was investigated using several configurations. Results regarding cavity shape and closure, as well as ventilation requirements versus cavitation number and Froude number are presented. In addition, 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. An attempt was made to correlate results from water tunnel experiments to open flows, where there are no effects of flow choking. Supercavitation parameters, especially the minimum attainable cavitation number are found to be strongly affected by tunnel blockage and Froude number.

[1]Sponsored by the Office of Naval Research

2:23PM D3.00002 Scaling of Partial Cavity Drag Reduction[1]
, STEVEN CECCIO, SIMO MAKIHARJU, SARAH SCHINASI, MARC PERLIN, University of Michigan — Experiments to examine ventilated partial cavities configured for drag reduction were conducted at the U.S. Navy's W. B. Morgan Large Cavitation Channel (LCC) and at the University of Michigan's 1:14th scale model of the LCC. Experiments were focused on the gas flux required to form and maintain stable partial cavities. The models and free surface forming gates were geometrically similar, and the experiments were performed over the same range of Froude numbers. The Reynolds numbers based on downstream distance were up to 80 million for the large and 1.8 million for the small-scale experiments. And, during the small-scale experiments the Weber number was varied by a factor of two to assess the effect of surface tension on the required air flux. The measured air fluxes normalized by flow speed, model span and step height varied by one to two orders of magnitude with the change of Reynolds number, and this is primarily due to the different gas entrainment process at the cavity closure. Variation of the Weber number at the small scale modestly changed the required gas fluxes.

[1]The research was sponsored by the Office of Naval Research under grant N00014-08-1-0215, Dr. L. P. Purtell program manager.

2:36PM D3.00003 Numerical Simulation of Cavitating Flows
, ASWIN GNANASKANDAN, KRISHNAN MAHESH, University of Minnesota — We are developing the capability to simulate cavitating flows (e.g sheet to cloud cavitation transition) in complex geometries. The compressible flow solver (Park & Mahesh, AIAA Paper 2007-0722) has been extended to solve for multiphase flows on unstructured meshes. A multiphase medium is constructed using a homogenous equilibrium model that assumes thermal equilibrium between the liquid phase and the vapor phase. The algorithm solves the compressible Navier Stokes Equations for the liquid/vapor medium along with the transport equation for the liquid's mass fraction. A characteristic-based shock capturing scheme is extended to handle non-ideal gases and mixtures, and applied in a predictor-corrector approach. The base scheme is non-dissipative and this approach ensures that the shock-capturing is active only in the regions of discontinuity. We will present details of this algorithm, its implementation, and validation.

2:49PM D3.00004 Large Eddy Simulation of cavitating turbulent flows
, SERGEI CHUMAKOV, Robert Bosch LLC, DAVID COOK, FRANK HAM, UWE IBEN — Large Eddy Simulation of a turbulent cavitating flow has been performed using the explicit spatially-filtered compressible Navier-Stokes solver Charles. The unstructured finite volume method uses a blended central-upwind scheme in single-phase regions to minimize artificial damping of resolved turbulence scales and switches to a lower-order reconstruction and an HLLC approximate Riemann solver to capture discontinuities associated with the phase change. Time discretization is performed with an explicit third order Runge Kutta scheme. Comparison between the simulations results and classic 1-D Riemann problems with and without cavitation are presented, as well as comparison with the cavitating flow experiments from the current literature.

3:02PM D3.00005 Cavitation inception criteria for hydrokinetic turbine blades
, IVAYLO NEDYALKOV, MARTIN WOSNIK, University of New Hampshire — Cavitation can adversely affect the performance of hydrokinetic turbines, and cause noise, vibration and even erosion. In some cases, unstable operation can be caused by cavitation-induced flow instabilities. A theoretical model for cavitation inception criteria on the blades of hydrokinetic turbines was developed by deriving cavitation numbers using turbine momentum theory and Airy wave theory. The cavitation number on a turbine blade element is calculated as a function of tip speed ratio, axial and angular induction factors at the rotor - which depend on the turbine's operating condition - and location on the blade, blade rotation angle, free stream velocity, wave-induced pressure oscillation, wave-induced velocities, time-dependent turbine hub submergence, vapor pressure, and free stream turbulence. With cavitation maps for specific hydrofoil shapes, which exist for some basic foil shapes, or can be obtained from inexpensive experiments in a small high-speed water tunnel where velocity and pressure can be controlled independently, the physical cavitation inception limits σ_i and $(\sigma/2\alpha)_i$ can be determined. With this model, safe deployment depths and safe tip speed ratios for specific turbines installed at a given site can be predicted. The model is compared to the common cavitation inception scaling with lift coefficient or Reynolds number.

3:15PM D3.00006 General linear theory for sound waves accompanied with evaporation and condensation , MASASHI INABA, Division of Mechanical and Space Engineering, Hokkaido University, TAKERU YANO, Department of Mechanical Engineering, Osaka University, MASAO WATANABE, Division of Mechanical and Space Engineering, Hokkaido University — When a sound wave in a vapor is reflected at an interface between the vapor and its liquid, the evaporation and condensation occurs in addition to the partial penetration of the wave into the liquid. Although the sound propagation outside the boundary layer is as usual governed by the linear wave equation, the boundary condition at the interface should carefully be treated in the kinetic theory of gases even if the Knudsen number (Kn) defined by the ratio of the mean free path of gas molecules to a typical wavelength is sufficiently small compared with unity. We therefore execute a general analysis of the entire flow field by using the ES-BGK model of the Boltzmann equation applicable to polyatomic gases and a general kinetic boundary condition at the interface. As a result of the asymptotic analysis for small Kn with the assumption that an acoustic Mach number is sufficiently small compared with Kn, we retrieve the linearized Euler equations outside the boundary layer and the so-called slip boundary conditions for the Euler and boundary layer equations. The coefficients in the slip conditions are determined by the Knudsen layer analysis. The effect of phase changes on the waves is illustrated by a simple example of plane standing wave.

3:28PM D3.00007 Fluid-structure interaction response and stability of flexible hydrofoils in cavitating flow , YIN LU YOUNG, ANTOINE DUCOIN, EUN JUNG CHAE, University of Michigan — There is an increasing interest in the use of passive/active control mechanisms to take advantage of the fluid-structure interaction response of flexible lifting bodies to improve propulsive efficiency and performance from ambient flow. However, design of these flexible lifting bodies are not trivial, particularly for heavily loaded and cavitating, off-design conditions, because of potential hydroelastic instability failure mechanisms such as divergence or flutter. Hence, the objectives of this research are to (i) develop and validate an efficient coupling procedure to predict the hydroelastic response of flexible hydrofoils in unsteady flows, and (ii) investigate the influence of fluid density and viscosity on the FSI response and stability of flexible hydrofoils in cavitating flows. A multiphase CFD code is coupled with a simplified 2-DOF model to represent the spanwise bending and twisting response of a flexible hydrofoil. The influence of coupling algorithms on the accuracy and stability of the numerical predictions are discussed.

3:41PM D3.00008 Three-dimensional investigation of the response of bubble/particle motion to grid generated turbulence , SAM RABEN, PAVLOS VLACHOS, Virginia Tech — Multiphase turbulent flow can be found in numerous different environments, from biological applications to industrial mineral processing. Often it is desirable to know how the discrete phase, bubbles and/or particles, respond to the turbulent fluctuations present in the surrounding flow. Previous works have developed empirical and semi- empirical relationships for the slip velocity, RMS motion, and collision rates for these discrete phases. These relationships were based upon 1- or 2-D samplings of the flow and therefore were approximations for the field as a whole. This work evaluates the validity of these relationships through the use of Tomographic PIV. Tomographic imaging it capable of providing not only information about the three-dimensional flow field but also motion of the discrete phases as well. An experiment was conducted in grid-generated turbulence where solid particles and bubbles where added. By fully resolving the motion of the flow field and objects present therein, the RMS and slip velocity could be calculated directly. This direct calculation was used to evaluate the validity of the currently available models.

3:54PM D3.00009 Influence of the bubbles on the turbulence in the liquid in hydrodynamic cavitation through a venturi , SYLVIE FUZIER, OLIVIER COUTIER DELGOSHA, SÉBASTIEN COUDERT, ANTOINE DAZIN, Laboratoire de Mécanique de Lille, France — The physical description of hydrodynamic cavitation is complex as it includes strongly unsteady, turbulent and phase change phenomena. Because the bubbles in the cavitation area render this zone opaque, nonintrusive experimental observation inside this zone is difficult and little is known about the detailed bubble, flow structure and physics inside. A novel approach using LIF-PIV to investigate the dynamics inside the cavitation area generated through a venturi is presented. The velocity in the liquid and of the bubbles are measured simultaneously and correlated with areas of various bubble structure. The influence of the bubble structure on the turbulence in the liquid is also studied.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D4 Drops II: Numerical Simulations 307

2:10PM D4.00001 Modeling and Numerical Predictions of the Dynamic Contact Angle Using a Microscopic Forcing Term , GERRY DELLA ROCCA, GUILLAUME BLANQUART, California Institute of Technology — The dynamic contact angle at the wall between two immiscible fluids has been shown to be a function of the static contact angle and the contact line velocity (or Capillary number). In numerical simulations of multiphase flows, this dynamic angle is often directly imposed from either an empirical model or a model based on the molecular-kinetic theory of wetting. In contrast, a hydrodynamic viewpoint is used here to describe the flow around the contact line, and a microscopic forcing term is applied to simulate the molecular interactions at the boundary. The numerical simulations rely on a staggered arrangement of velocities, and a level set method is used to track the interface location. This framework has an inherent amount of slip which can be further supplemented by a Navier-slip condition. For the pressure jump across the interface, a ghost fluid method is employed and the microscopic forcing term is added into the jump condition. An unresolved curvature model is proposed for the microscopic forcing term where a smooth circular arc connects the dynamic and static angles in the sub- grid gap. This model does not rely on any underlying formula connecting the contact angles and the contact line velocity, but rather the dynamic angle evolves due to the nature of the flow. The model predictions for the dynamic contact angle are compared to experimental results.

2:23PM D4.00002 Numerical Simulation and Scaling Analysis of Cell Printing , RUI QIAO, PING HE, Clemson University — Cell printing, i.e., printing three dimensional (3D) structures of cells held in a tissue matrix, is gaining significant attention in the biomedical community. The key idea is to use inkjet printer or similar devices to print cells into 3D patterns with a resolution comparable to the size of mammalian cells. Achieving such a resolution in vitro can lead to breakthroughs in areas such as organ transplantation. Although the feasibility of cell printing has been demonstrated recently, the printing resolution and cell viability remain to be improved. Here we investigate a unit operation in cell printing, namely, the impact of a cell-laden droplet into a pool of highly viscous liquids. The droplet and cell dynamics are quantified using both direct numerical simulation and scaling analysis. These studies indicate that although cell experienced significant stress during droplet impact, the duration of such stress is very short, which helps explain why many cells can survive the cell printing process. These studies also revealed that cell membrane can be temporarily ruptured during cell printing, which is supported by indirect experimental evidence.

2:36PM D4.00003 ABSTRACT WITHDRAWN —

2:49PM D4.00004 Confined shear induces spatial ordering in an interacting pair of drops[1] , RAJESH SINGH, KAUSIK SARKAR, University of Delaware — In a shear flow between two parallel walls, viscous interactions between deforming drops induce migration velocities away from the wall. Unlike free shear, the drops, after collision, come to the centerline of the domain, i.e. they experience a zero cross-stream separation, and more importantly a fixed streamwise separation. We numerically investigate to find that the final streamwise separation depends on the degree of confinement, capillary number and viscosity ratio, but does not depend on the initial separation between drops. The streamwise separation varies as the square of the domain size and inversely with the capillary number. We present an asymptotic theory to explain these scalings.

[1]Partially supported by NSF

3:02PM D4.00005 Breakup of a Liquid Drop Falling Through a Quiescent Media: A DNS Study , MAZIYAR JALAAL, KIAN MEHRAVARAN, School of Engineering, University of British Columbia, 3333 University Way, Kelowna, V1V 1V7 BC, Canada — The breakup of falling liquid droplets in a stationary media is studied using Direct Numerical Simulations (DNS). An adaptive Volume of Fluid (VOF) method based on an octree Cartesian grid generation is employed, considerably reducing the computational cost. Fragmentations are followed reaching approximately stable clouds of droplets up to 1/1000 of the initial droplet diameter. Three different simulations are performed investigating the influence of the initial Eötvös number. The mechanism of breakup, one of the most unclear phenomena in multiphase systems is described in detail. The wave growth over the bag, creation and retraction of punctures and ligament formation are presented and results are compared with recent theoretical investigations of (*Savva & Bush, J. Fluid. Mech.* **626***: 211-240.*) and (*Bermond & Villermaux, J. Fluid. Mech.* **524***: 121-130.*). The roles of Rayleigh-Taylor and Rayleigh-Plateau instabilities on breakup are also described and their influences on further cluster-of-fragments creation are shown. The outcomes can be used to develop current secondary atomization models. Moreover, the results can be used for better understanding of rain drop atomization during precipitation, as well as water droplet atomization in cooling towers.

3:15PM D4.00006 Bare Shear Viscosity and Anomalous Fall Rate of Oil Droplets in Nitrogen , RODNEY VARLEY, Hunter College C.U.N.Y. — Experimental evidence of Kim and Fedele (1982) indicates a breakdown of the Millikan Law for the fall rate of oil droplets in Nitrogen gas over a pressure range of 1-15 atm. The discrepancy is most pronounced for smallest, 0.1 micron radius droplets for which the fall rate increases with pressure. The opposite behavior was observed by Millikan with larger drops in air of pressure at most one atm. We explain these results by arguing that the particle's motion, in particular Stokes' drag formula, is determined by the so-called bare shear viscosity which applies to micro fluid flows. This is in contrast with the usual theory which uses a renormalized shear viscosity and which is well approximated by the Enskog value. A mode coupling formula for the bare shear viscosity is discussed and a graphical comparison is made with the experimental results. Basically an increase in gas pressure produces a decrease in the bare shear viscosity and thus the fall rate increases. The idea that the shear viscosity is smaller for micro flows is consistent with the intuitive belief that on small enough spatial and time scales, fluid flows are conservative without dissipation.

3:28PM D4.00007 Fluid Dynamics of Bottle Filling , PATRICK MCGOUGH, HAIJING GAO, SANTOSH APPATHURAI, OSMAN BASARAN, Purdue University — Filling of bottles is a widely practiced operation in a large number of industries. Well known examples include filling of "large" bottles with shampoos and cleaners in the household products and beauty care industries and filling of "small" bottles in the pharmaceutical industry. Some bottle filling operations have recently drawn much attention from the fluid mechanics community because of the occurrence of a multitude of complex flow regimes, transitions, and instabilities such as mounding and coiling that occur as a bottle is filled with a fluid. In this talk, we present a primarily computational study of the fluid dynamical challenges that can arise during the rapid filling of bottles. Given the diversity of fluids used in filling applications, we consider four representative classes of fluids that exhibit Newtonian, shear-thinning, viscoelastic, and yield-stress rheologies. The equations governing the dynamics of bottle filling are solved either in their full 3D but axisymmetric form or using the slender-jet approximation.

3:41PM D4.00008 Do Liquid Drops on Inclined Surfaces Slide or Roll? , SUMESH PT, Jawaharlal Nehru Centre for Advanced Scientific Research, Bangalore, IGNACIO PAGONABARRAGA, Department de Fisica Fonamental, Universitat de Barcelona, RONOJOY ADHIKARI, The Institute of Mathematical Sciences, Chennai, RAMA GOVINDARAJAN, Jawaharlal Nehru Centre for Advanced Scientific Research, Bangalore — A solid sphere is likely to roll, while a rectangular box is likely to slide, on an inclined surface. Instead, a liquid drop can exhibit a variety of shapes and complex but interesting dynamics. We obtain global minimum energy static shapes first, for two realistic bases of potential energy, front and back-pinned. We find that the free end always assumes Young's equilibrium angle. Using this clue, simple equations describing the angles and the maximum volume may be derived. Combining the lattice Boltzmann method for hydrodynamics and method of lines for a Cahn-Hilliard equation, a hybrid numerical scheme is developed to study the dynamics of binary fluids on an inclined plate. The contribution of pure translation, and the vorticities associated with rolling and shearing motion are distinguished, using which the motion of the drop can be split into roll and slip. Surprisingly, as gravity increases, the fraction of motion due to roll decreases significantly for certain contact angles. The rolling motion is strongly dependent on the slip length which is in contrast to predictions by the lubrication approximation, where all dependence on the slip length is generally logarithamic.

3:54PM D4.00009 Repellency of the Lotus Leaf: Resistance of Water Intrusion under Hydrostatic Pressure , C.W. EXTRAND, Entegris — In an attempt to better understand the repellency of the lotus leaf, a model was constructed from hydrophobic hemispheres arranged on a hexagonal array. Two scenarios were considered. In the first, the hemispheres were smooth. In the second, the hemispheres had a secondary roughness. The model shows that without the secondary structure, the repellency of this surface geometry is relatively poor. The secondary structure directs the surface tension upward, allowing much greater resistance to penetration of water and prevents the loss of repellency. From the proposed model, the maximum intrusion pressure (or so-called Cassie-Wenzel transition) of the lotus leaf is estimated to be 12-15 kPa. The predicted maximum pressure agrees well with reported values from experimental measurements.

4:07PM D4.00010 Surface and Bulk Oscillations of Sessile Drops: Clearing Up Confusion and Understanding Wind Sheared Drops , ANDREW J.B. MILNE, University of Alberta, BEATRIZ DEFEZ GARCIA, Universidad Politecnica de Valencia, MIGUEL CABRERIZO VILCHEZ, Universidad de Granada, ALIDAD AMIRFAZLI, University of Alberta — Sessile drop oscillations are studied in the presence of a shearing airflow, and varying body force. The various possibilities for analysis, (center of mass or drop surface oscillations) are elucidated through presenting a unifying analysis framework based on wavenumber, frequency, and fluid properties. This work examines a range of fluid properties in a single study for the first time. A dispersion relation is found relating the frequency of centroid oscillation and capillary-gravity wave number, depending on the ratio (surface tension/liquid density)$^{1/2}$, drop size$^{-3/2}$ and contact angle. The effects of contact angle are more complex than previously suggested simplifications, or analytic solutions for axisymetric drops and must at present be treated empirically. The growth of sessile drop oscillations is linear at low air velocities and exponential at higher air velocities. This is explained by drawing analogies to drops experiencing a varying body force, and to wind driven capillary-gravity waves on lakes, respectively. Liquid viscosity retards the growth of the waves, and has other important effects.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D5 CFD II: High-order Methods 308

2:10PM D5.00001 High order DG discretizations with p-adaptive limiting for high speed flows on mixed type meshes, JOHN EKATERINARIS, KONSTANDINOS KONTZIALIS, School of Mechanical and Aerospace Eng., University of Patras, Gereece — High order accurate discontinuous Galerkin (DG) discretizations are applied for mixed-type meshes in two and three dimensions. Collapsed coordinate transformations are used and all calculations of DG discretizations for arbitrary mixed-type meshes are performed in the canonical computational space. High order accurate implicit time marching is used for time advancement. For computations of high speed flows with strong discontinuities, a new limiting approach that is applied in the canonical computational domain and it is suitable for arbitrary-type meshes, is employed. Limiting is applied with a modified total variation bounded (TVB) limiter, previously used for DG discretizations in quadrilateral meshes. Accurate evaluation of the variation of the second derivative of the approximate solution allows to accurately detect regions of discontinuities where limiting must be applied and progressively increase the accuracy of the computed solution away from discontinuities. For the computation of flows with large pressure and density ratios, positivity preserving limiters are used to ensure that density and pressure remain well defined. Application of the new limiting approach and positivity preserving limiters to a number of standard test cases of high speed flows with discontinuities is carried out to demonstrate the versatility and robustness of the proposed method.

2:23PM D5.00002 High-Order Finite-Difference Solution of the Poisson Equation Involving Complex Geometries in Embedded Meshes, ALEXANDRE MARQUES, Massachusetts Institute of Technology, JEAN-CHRISTOPHE NAVE, McGill University, RUBEN ROSALES, Massachusetts Institute of Technology — The Poisson equation is of central importance in the description of fluid flows and other physical phenomena. In prior work, Marques, Nave, and Rosales introduced the Correction Function Method (CFM) to obtain fourth-order accurate solutions for the constant coefficient Poisson problem with prescribed jump conditions for the solution and its normal derivative across arbitrary interfaces. Here we combine this method with the ideas introduced by Mayo to solve other Poisson problems involving complex geometries. In summary, we are able to rewrite the problem as a boundary integral equation in terms of a potential distribution over the boundary or interface. The solution of this integral equation is discontinuous across the boundary or interface. Hence, after this integral equation is solved using standard techniques, the potential distribution can be used to determine the jump discontinuities. We are then able to use the CFM to solve the resulting Poisson equation with jump discontinuities. The outcome is a fourth-order accurate scheme to solve general Poisson problems which, over arbitrary geometries, has a cost that is approximately twice that of a fast Poisson solver using FFT on a rectangular geometry of the same size. Details of the method and applications will be presented.

2:36PM D5.00003 Polynomial Interpolation of 12^{th}-Order Combined Compact Difference Schemes onto Non-Uniform Grids in Numerical Simulations of Boundary Layer Turbulence Transition, J.C. CHEN, WEIJIA CHEN, Nanyang Technological University — Numerical simulations of boundary layer turbulence transition meet with the challenge of astronomical computational demands needed to visualize the underlying physical phenomena. Reprieve from the onerous computational load avails when recognizing that boundary layer turbulent flow structures intermittently concentrate near the wall region. Astute implementation of non-uniform computational grids with microscopically precise resolution near the wall boundary that tapers off away from the wall offers potential for vast computational efficiency. Importantly, the numerical method applied on non-uniform grids must further consider the issue of numerical stability of the numerical schemes at the wall. Implementation of a piece- wise non-uniform grid accounts for both the numerical demands of realizing turbulent flow structures near the wall and maintaining numerical stability of the wall boundary schemes. Customization of a 12^{th} order Combined Compact Difference Scheme with 10^{th} and 11^{th} order boundary schemes to the non-uniform grid by polynomial interpolation enhances the numerical accuracy near the wall over the use of coordinate transformation. Analyses of the eigenvalue spectrum and modified wave number intimate the stability of the numerical method.

2:49PM D5.00004 An Eigen-based Spectral Element Method[1], SUCHUAN DONG, XIAONING ZHENG, Purdue University — We present an efficient high-order spectral element method. The method employs a novel numerically-constructed expansion basis within an element, which represents an optimal set of basis functions and simultaneously diagonalizes the mass and stiffness operators. We compare the new method with the Jacobi polynomial-based spectral element method that is commonly used in computational fluid dynamics. Results demonstrate that the new method enjoys a considerably superior numerical efficiency in terms of numerical conditioning and the number of iterations to convergence for iterative solvers.

[1]Supports from ONR and NSF are gratefully acknowledged.

3:02PM D5.00005 An approach to satisfy pressure and temperature equilibriums at interfaces in compressible multicomponent flows using high-order schemes, HIROSHI TERASHIMA, University of Tokyo, SOSHI KAWAI, Institute of Space and Astronautical Science of JAXA, MITSUO KOSHI, University of Tokyo — We present a formulation for high-order simulations of compressible multicomponent flows using a sixth-order compact differencing scheme and a localized artificial diffusivity. The formulation is designed to satisfy both of pressure and temperature equilibriums at fluid interfaces by introducing additional two equations to the Euler equations. In order to deal with sharp initial condition of density, a localized artificial diffusivity term is introduced to the mass conservation equation. Several one-dimensional problems such as advection of contact and material interfaces and a shock tube problems demonstrate that the present method maintains the pressure and temperature equilibriums and also satisfies the mass conservation property. The localized artificial diffusivity for the mass conservation equation enables to start computations even with severe one-point jump condition, effectively reducing numerical wiggles at the fluid interfaces. Comparisons with a conventional full conservative formulation present the superiority of the present method for preventing spurious pressure/velocity/temperature oscillations at the fluid interfaces. Two-dimensional problems such as the Richtmyer-Meshkov instability demonstrate its multidimensional applicability.

3:15PM D5.00006 A Quadrilateral Spectral Multidomain Penalty Method Model For High Reynolds Number Incompressible Stratified Flows, JORGE ESCOBAR-VARGAS, PETER DIAMESSIS, Cornell University — We present a spectral multidomain penalty method-based incompressible Navier Stokes solver for high Reynolds number stratified turbulent flows in doubly non-periodic domains. Within the solver, time is discretized with a fractional-step method, and, in space, a Gauss-Lobatto-Legendre collocation approach is used in discontinuous quadrilateral subdomains. Stability of the numerical scheme is guaranteed through a penalty scheme and spectral filtering, further buttressed by a overintegration-based dealiasing technique. The efficient iterative solution of the associated discrete pressure Poisson equation is ensured through a Kronecker product based computation of the null vector associated with the global matrix, plus a two-level preconditioner within a GMRES solver. Efficiency and accuracy of the Navier Stokes solver are assessed through the solution of the lid-driven cavity flow, Taylor vortex and double shear layer. The canonical lock exchange problem is also presented to assess the potential of the solver for the study of environmental stratified flows.

3:28PM D5.00007 Spectral element simulations of unsteady flow over a 3D, low aspect-ratio semi-circular wing, SRIHARSHA KANDALA, DIETMAR REMPFER, IIT, Chicago — Numerical simulations of unsteady 3D flow over a low-aspect-ratio semi-circular wing are performed using a spectral element method. Specsolve, a parallel spectral element solver currently under development at IIT, is used for the simulation. The solution is represented locally as a tensor product of Legendre polynomials and C^0-continuity is enforced between adjacent domains. A BDF/EXT scheme is used for temporal integration. The fractional step method is used for computing velocity and pressure. The code incorporates a FDM (fast diagonalization method) based overlapping Schwarz preconditioner for the consistent Poisson operator and an algebraic multigrid based coarse grid solver (P.F. Fischer et al, J. Phys.: Conf. Ser.(125) 012076, 2008) for pressure. The simulation replicates the conditions of the active flow control experiment (D. Williams et al, AIAA paper, 2010-4969). The Reynolds number based on chord length and free-stream velocity is about 68000. Different angles of attack, encompassing both pre-stall and post-stall regimes, are considered. These results are compared with data from the experiment and numerical simulations based on Lattice Boltzmann method (G. Brès et al, AIAA paper, 2010-4713).

3:41PM D5.00008 A Computational Study of Transient Couette Flow Over an Embedded Cavity Surface[1] , MICHAEL THOMPSON, Arizona State University, AMY LANG, WILL SCHREIBER, CHASE LEIBENGUTH, JOHN PALMORE, Univeristy of Alabama — Insect flight has become a topic of increased study due to bio-inspired applications for Micro-Air-Vehicles (MAVs). The complex yet efficient flight mechanism of butterflies relies upon flexible, micro-geometrically surface patterned, scaled wings. Effective vortex control, when flapping as well as low-drag gliding, may result from the wing's texture. This hypothesis was tested by focusing on the formation of embedded vortices between the rows of scales on butterfly wings. To calculate the total surface drag induced on the moving cavity surface a computational fluid dynamics study using FLUENT simulated the flow inside and above the embedded cavities under transient Couette flow conditions with Reynolds numbers varied from 0.01 to 100. The computational model consisted of a single embedded cavity with a periodic boundary condition. Based on SEM pictures of Monarch (*Danaus plexippus*) butterfly scales, various cavity geometries were tested to deduce drag reduction. Results showed that the embedded vortex size and shape generated within the cavity depended on which surface moved (top, flat wall or bottom, cavity wall) as well as aspect ratio. Surface drag reduction was confirmed over the cavity surfaces when compared to that of a flat plate, and increased with aspect ratio.

[1]Funded by REU SITE EEC - 1062611

3:54PM D5.00009 Computational Analysis of Low Reynolds Number Couette Flow Over an Embedded Cavity Geometry[1] , CHASE LEIBENGUTH, AMY LANG, WILL SCHREIBER, The University of Alabama — A butterfly utilizes an efficient and complex flight mechanism comprised of multiple interacting flow control devices that include flexible, micro-geometrically surface patterned, scaled wings. The following research attempts to deduce any aerodynamic advantages that arise from the formation of vortices in between successive rows of scales on a butterfly wing. The simplified computational model consists of an embedded cavity within a Couette flow with the flat plate moving transversally over the cavity. The effects of cavity geometry and Reynolds number are analyzed separately. The model is simulated in ANSYS FLUENT and provides qualitative insight into the interaction between the scales and the boundary layer. Preliminary results indicate that vortices form within the cavity and potentially contribute to a net partial slip condition. Further, the embedded cavities contribute to a net reduction in the drag coefficient that varies with Reynolds number and cavity geometry.

[1]Funded by NSF REU Grant EEC - 0754117

4:07PM D5.00010 A full Navier-Stokes solver on Irregular Domains coupled with a Poisson-Boltzmann solver with Neumann or Robin boundary conditions on Non-Graded Adaptive Grid , ASDIS HELGADOTTIR, FREDERIC GIBOU, University of California Santa Barbara — We introduce a second-order solver for the full Navier-Stokes equations coupled with the Poisson-Boltzmann equation on irregular domains. This simple fluid solver can be used for simulating fluid flow in microfluidic devices. The irregular domain is described implicitly and the grid needs not to conform to the domain's boundary, which makes grid generation straightforward and robust. Finite Volume approach is used for the Poisson-Boltzmann solver making it straightforward to enforce Neumann or Robin boundary conditions at the irregular domain. The linear system is symmetric, positive definite in the case where the grid is uniform, otherwise nonsymmetric but an invertible M-matrix. The fluid solver is based on the projection method where finite volume approach is used to easily enforce Neumann boundary conditions for pressure at the irregular domain. Extensions from two spatial dimensions to three are straightforward both for the Poisson-Boltzmann solver and the Navier-Stokes solver. The Poisson-Boltzmann solver is solved using Quadtee/Octree grids (in two and three spatial dimensions respectively). The fluid solver is solved on uniform grid and values are interpolated between the two grids using second order accurate interpolation.

Sunday, November 20, 2011 2:10PM - 4:07PM –
Session D6 Insect Flight II 309

2:10PM D6.00001 How mosquitoes fly in the rain , ANDREW DICKERSON, PETER SHANKLES, NIHAR MADHAVAN, DAVID HU, Georgia Institute of Technology — Mosquitoes thrive during rainfall and high humidity. If raindrops are 50 times heavier than mosquitoes, how do mosquitoes fly in the rain? In this combined experimental and theoretical study, we measure the impact force between a falling drop and a free-flying mosquito. High-speed videography of mosquitoes and custom-built mimics reveals a mosquito's low inertia renders it impervious to falling drops. Drops do not splash on mosquitoes, but simply push past them allowing a mosquito to continue on its flight path undeterred. We rationalize the force imparted using scaling relations based on the time of rebound between a falling drop and a free body of significantly less mass.

2:23PM D6.00002 The Timing in the Control of Insect Flight Instability , SONG CHANG, Applied and Engineering Physics, Cornell University, Z. JANE WANG, Mechanical and Aerospace Engineering, Cornell University — Flapping flight of insects is intrinsically unstable. Using 3D dynamic simulation of flapping flight, we analyze the stability of periodic states associated with the limit cycles of the dynamical system. We construct a discrete time-delayed linear controller and examine the controllability condition. The controller's effectiveness depends in a subtle manner on the timing of the sensory measurement combined with the delay time in actuation.

2:36PM D6.00003 Uncovering the aerodynamics of the smallest insects using numerical and physical models , LAURA MILLER — 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*. The smallest flying insects have received far less attention, although previous work has shown that flight kinematics and aerodynamics can be significantly different. In this presentation, three-dimensional direct numerical simulations are used to compute the lift and drag forces generated by flexible wings to reveal the aerodynamics of these tiny fliers. Results are validated against dynamically scaled physical models. At the lowest Reynolds numbers relevant to insect flight, the relative forces required to rotate the wings and fling them apart become substantially greater. Wing flexibility can reduce these forces and improve efficiency in some situations.

2:49PM D6.00004 Flow Structure on a Flapping Wing: Quasi-Steady Limit , CEM OZEN, DONALD ROCKWELL, Lehigh University — The flapping motion of an insect wing typically involves quasi-steady motion between extremes of unsteady motion. This investigation characterizes the flow structure for the quasi-steady limit via a rotating wing in the form of a thin rectangular plate having a low aspect ratio (AR =1). Particle Image Velocimetry (PIV) is employed, in order to gain insight into the effects of centripetal and Coriolis forces. Vorticity, velocity and streamline patterns are used to describe the overall flow structure with an emphasis on the leading-edge vortex. A stable leading-edge vortex is maintained over effective angles of attack from 30° to 75° and it is observed that at each angle of attack the flow structure remains relatively same over the Reynolds number range from 3,600 to 14,500. The dimensionless circulation of the leading edge vortex is found to be proportional to the effective angle of attack. Quasi-three-dimensional construction of the flow structure is used to identify the different regimes along the span of the wing which is then complemented by patterns on cross flow planes to demonstrate the influence of root and tip swirls on the spanwise flow. The rotating wing results are also compared with the equivalent of translating wing to further illustrate the effects of the rotation.

3:02PM D6.00005 Computational Aerodynamics of Insects' Flapping Flight , KYUNG DONG YANG, Blair Academy, RICHARD KYUNG, Seoul National University — The kinematics of the Insects' flapping flight is modeled through mathematical and computational observations with commercial software. Recently, study on the insects' flapping flight became one of the challenging research subjects in the field of aeronautics because of its potential applicability to intelligent micro-robots capable of autonomous flight and the next generation aerial-vehicles. In order to uncover its curious unsteady characteristics, many researchers have conducted experimental and computational studies on the unsteady aerodynamics of insects' flapping flight. In the present paper, the unsteady flow physics around insect wings is carried out by utilizing computer software e-AIRS. The e-AIRS (e-Science Aerospace Integrated Research System) analyzes and models the results of computational and experimental aerodynamics, along with integrated research process of these two research activities. Stroke angles and phase angles, the important two factors in producing lift of the airfoils are set as main parameters to determine aerodynamic characteristics of the insects' flapping flight. As a result, the optimal phase angle to minimize the drag and to maximize the lift are found. Various simulations indicate that using proper value of variables produce greater thrust due to an optimal angle of attack at the initial position during down stroke motion.

3:15PM D6.00006 Butterfly scales and their local surface drag dependence on flow orientation , AMY LANG, ROBERT JONES, University of Alabama — An experimental study was carried out to measure surface drag over embedded cavity models based on the geometry of butterfly wing scales. Monarch (*Danaus plexippus*) scales, each measuring about 0.1 mm in length, were observed using microscopy to evaluate the microgeometry. Two separate, fabricated models scaled up (300:1) the geometry for dynamically similar testing in a Couette flow oil tank facility. The drag induced over the patterned surfaces was measured using a force gauge. Flow transverse to the rows of scales resulted in a significant drag decrease ($>$ 30%), with dependence on Re. This drag reduction is attributed to the formation of embedded vortices forming between the rows of scales resulting in a "roller bearing" effect. Flow parallel to the rows, as expected, resulted in larger drag increases, especially at lower Re. Both effects may prove beneficial to the butterfly, during flapping and gliding flight, and will be discussed based on the observed orientation of the scales on real specimens.

3:28PM D6.00007 The importance of being top-heavy: Intrinsic stability of flapping flight , LEIF RISTROPH, New York University, BIN LIU, Brown University, JUN ZHANG, New York University — We explore the stability of flapping flight in a model system that consists of a pyramid-shaped object that freely hovers in a vertically oscillating airflow. Such a "bug" not only generates sufficient aerodynamic force to keep aloft but also robustly maintains balance during free-flight. Flow visualization reveals that both weight support and intrinsic stability result from the periodic shedding of dipolar vortices. Counter-intuitively, the observed pattern of vortex shedding suggests that stability requires a high center-of-mass, which we verify by comparing the performance of top- and bottom-heavy bugs. Finally, we visit a zoo of other flapping flyers, including Mary Poppins' umbrella, a flying saucer or UFO, and Da Vinci's helicopter.

3:41PM D6.00008 Vortex Tilting and the Enhancement of Spanwise Flow in Flapping Flight , SPENCER FRANK, University of Central Florida, GIOVANNI BARBERA, BO CHENG, XINYAN DENG, Purdue University — The leading edge vortex is key in lift generation on flapping wings. Its stability depends on the transport of the entrained vorticity into the wake via spanwise flow. This study investigates the generation and enhancement of spanwise flow based on the chordwise vorticity that results from the tilting of the leading edge vortex and trailing edge vortex. Two dynamically scaled robotic model wings, one rectangular and one insect wing shaped based on *Drosophila melanogaster* (fruit fly), are submerged in a tank of mineral oil and actuated into flapping motion. The overall flow structure was visualized and measured by a Volumetric 3-component Velocimetry (V3V) system (TSI, Inc.). From the three dimensional flow measurements obtained, the chordwise vorticity resulting from the vortex tilting is shown. The distribution of the resulting spanwise flow induced by the vortex tilting is shown using isosurfaces and on a planar cross section downstream of the leading edge. It is observed that the largest spanwise flow is located in the area between the tilted leading edge vortex and the tilted trailing edge vortex, supporting our hypothesis that the vortex tilting enhances the spanwise flow. This vortex tilting mechanisms helps to explain the overall flow structure and the stability of the leading edge vortex.

3:54PM D6.00009 Simulation of a prescribed fruitfly flapping motion by the ALE-GFD method on a hybrid Cartesian-meshfree grid , N.T. TRONG, T.T. LIM, K.S. YEO, National University of Singapore — The flapping-wing aerodynamics of insects has been a subject of immense interest for many years. The Arbitrary Lagrangian-Eulerian – Generalized Finite Difference (ALE-GFD) computational scheme on hybrid convecting Cartesian-meshfree grid systems represents a viable alternative to existing mesh-based and immersed boundary approaches for simulating the highly complex and unsteady flows generated by the flapping wings of insects. The three-dimensional flapping-wing flows of a fruitfly (Drosophila) are simulated in the present study. As the forces generated are very sensitive to the acceleration of the wings, a smoothing process was applied on the flapping kinematics to suppress non-physical fluctuations and spikes from the force outcome. The resulting lift and drag forces are then validated with the experimental results, obtained from a parallel experimental study conducted by the research group, measured on a fruitfly-like wing profile executing the identical motion. The excellent agreement between the results demonstrates the feasibility and efficacy of the ALE-GFD numerical approach.

Sunday, November 20, 2011 2:10PM - 4:07PM –
Session D7 Turbulent Boundary Layers II: Roughness 310

2:10PM D7.00001 Effect of surface finish on the onset of roughness effects in turbulent channel flow[1] , MICHAEL SCHULTZ, KAREN FLACK, United States Naval Academy — The ability to reliably predict the onset of roughness effects in wall-bounded turbulent flows is an important yet elusive aim. This is especially relevant for the test engineer or experimental researcher who must specify the surface finish for a wind or water tunnel model. In this study, the effect of the surface finish resulting from sanding was investigated systematically using a turbulent channel flow facility. The experiments were conducted over a Reynolds number range of 6,000 – 64,000 based on the channel height and the bulk mean velocity. Painted surfaces sanded with a range of grit sizes were tested. The results indicate that all of these surfaces depart from hydraulically smooth behavior at roughness Reynolds numbers (k^+_{rms}) of between 0.5 and 0.7. The implications of these results with regard to model surface finish required in order to avoid roughness effects will be discussed.

[1]This work was funded by ONR.

2:23PM D7.00002 Important scales for predicting the onset of roughness effects in the transitionally rough regime[1] , KAREN FLACK, MICHAEL SCHULTZ, U S Naval Academy — Accurately predicting the departure from hydraulically smooth behavior for a given surface has significant engineering applications including determining the frictional drag of vehicles or the pressure drop in piping systems. A series of experiments have been performed for a range of three-dimensional rough surfaces to determine the appropriate roughness scales for the prediction of the onset of roughness effects. The experiments were performed in a small water channel with a Reynolds number based on channel height range of 6,000-64,000. The wall shear stress was determined from the pressure drop in the in the channel. Scaling parameters based on the statistics of the surface topography in addition to the roughness density will be presented.

[1]This work was supported by the Office of Naval Research

2:36PM D7.00003 Direct numerical simulation of a high-entrainment turbulent boundary layer[1] , GUILLEM BORRELL, U. Politecnica Madrid, AYSE G. GUNGOR, JAVIER JIMENEZ

— It has been reported that certain rough surfaces modify the outer region of turbulent boundary layers, but not those of channels or pipes. Besides their surface geometries, all those experiments share relatively large spreading and entrainment rates, which is known to modify the outer intermittent layers of external turbulent flows, but is absent from channels. To separate the effect of surface geometry from that of entrainment, we present a direct simulation of a zero-pressure-gradient turbulent boundary layer, at $Re_\theta = 1400 - 4500$, in which the friction coefficient is augmented by a smooth volumetric force, restricted to the viscous layer below $y^+ = 25$, and proportional to the streamwise component of the velocity. The spreading rate increases by 70%, equivalent to a sand roughnes $k_s^+ \approx 60$. The resulting changes in the velocity and pressure fluctuations, and in the velocity correlation lengths, are compared with those of rough-wall experiments.

[1]Funded by CICYT and PRACE.

2:49PM D7.00004 Investigation of wall-bounded turbulence over sparsely distributed roughness , MARCO PLACIDI, BHARATH GANAPATHISUBRAMANI, University of Southampton

— The effects of sparsely distributed roughness elements on the structure of a turbulent boundary layer are examined by performing a series of Particle Image Velocimetry (PIV) experiments in a wind tunnel. From the literature, the best way to characterise a rough wall, especially one where the density of roughness elements is sparse, is unclear. In this study, rough surfaces consisting of sparsely and uniformly distributed LEGO® blocks are used. Five different patterns are adopted in order to examine the effects of frontal solidity (λ_f, frontal area of the roughness elements per unit wall-parallel area), plan solidity (λ_p, plan area of roughness elements per unit wall-parallel area) and the geometry of the roughness element (square and cylindrical elements), on the turbulence structure. The Karman number, Re_τ, has been matched, at the value of approximately 2300, in order to compare across the different cases. In the talk, we will present detailed analysis of mean and rms velocity profiles, Reynolds stresses and quadrant decomposition.

3:02PM D7.00005 Non-equilibrium layers in rough-wall channel flow[1] , JONATHAN MORRISON, Imperial College, DAVID BIRCH, University of Surrey, UK

— Streamwise and wall-normal velocity statistics in a fully- developed turbulent channel flow over large relative roughness are examined, for $k/h = 3.9\%$, and $Re_\tau = 5130$. The surface is fully rough, $k^+ = 200$, and the appearance of a log region is demonstrated by an overlap region with inner and outer scaling, where the procedure defines a roughness lengthscale and a zero-plane displacement, $d^+ = 130$ for $\kappa = 0.41$. Although the zero-plane displacement is somewhat smaller than the roughness height, it is large relative to the thickness of the log region, $205 < (y-d)^+ < 308$. The zero-plane displacement obtained by the matching to the log law is somewhat smaller than that required for the measured shear-stress gradient to match the linear variation required by the momentum equation, and the latter does not provide a convincing log law. The reasons for this are discussed and the behaviour of the turbulence in the log region is examined, with particular reference to the maximum in the Reynolds shear stress which appears above the outermost extent of the log region. The effect of the roughness offset on the energy balance is examined: despite the appearance of the log law, it does not conform to the local-equilibrium approximation. Scaling of the velocity spectra and second- and third-order structure functions is examined: the energy balances in the log region and at the centre-line where estimates of the dissipation rate can be made are examined.

[1]EPSRC Grant EP/D037166

3:15PM D7.00006 DNS of Rough Surface Turbulent Boundary Layer[1] , JAMES CARDILLO, RPI, GUILLERMO ARAYA, Texas Tech University, YI CHEN, RPI, KENNETH JANSEN, University of Colorado-Boulder, ONKAR SAHNI, RPI, LUCIANO CASTILLO, Texas Tech University

— A dynamic method for prescribing realistic inflow boundary conditions is presented for simulations of spatially developing turbulent boundary layers subject to surface roughness. Direct Numerical Simulation (DNS) of a moderate Reynolds number, zero pressure gradient (ZPG) turbulent boundary layer was performed. The boundary layer was subjected to transitional, $24-$grit sand grain surface roughness, modeled with a roughness parameter of $k^+ \sim$12 and a Reynolds number of $R_\theta = 2400$. The computational method involves a synergy of the multi-scale dynamic approach and a new methodology for mapping high-resolution topographical surface data into a computational fluid dynamics environment. It is shown here that the multi-scale dynamic approach can be successfully extended to simulations, which incorporate surface roughness. In terms of the mean velocity and Reynolds stresses, the DNS results are encouraging as they demonstrate good agreement with the LDA measurements performed under similar conditions.

[1]NSF-CBET and ONR (R. Joslin)

3:28PM D7.00007 Direct Numerical Simulation of a turbulent channel flow with 3D wedges randomly placed on a wall , STEFANO LEONARDI, LUIS MARTINEZ TOSSAS, EDGARDO GARCIA CARTAGENA, University of Puerto Rico at Mayaguez

— Direct numerical simulations of a turbulent channel flow with 3D wedges of random height have been performed. In addition, two other simulations have been carried out to assess the effect of the geometry on the overlying flow. In the first simulation, the elements in the wake of higher wedges were removed while in the other, a uniform distribution of wedges with the same area was used. A wedge is considered in the wake of another element when the line joining the crests is steeper than 45 degrees. The bulk Reynolds numbers is 7000 which correspond in case of smooth walls to $Re_\tau = 300$. The comparison of the 3 surfaces has shown that near the wall, the uniformly distributed roughness represents only a poor approximation of the surface with wedges of random height. The surface obtained removing the wedges in the wake of previous wedges present Reynolds stresses, pressure distribution and spectra in good agreement with the original surface. Therefore, it is possible to reduce the geometrical complexity of a rough surface while retaining the same flow physics.

3:41PM D7.00008 Direct numerical simulation study in turbulent boundary layers with the cube-roughened walls[1] , JUNSUN AHN, JAE HWA LEE, HYUNG JIN SUNG, KAIST

— Direct numerical simulations of turbulent boundary layers over the cube-roughened walls were performed to investigate the effects of the streamwise (p_x/k) and spanwise (p_z/k) extents between the cubes on the properties in the outer-layer statistics. The streamwise extent of the cubes was varied from p_x/k=2, 3, 4, 6, 8, and 10 at fixed p_z/k=2 and, the spanwise extent of the cubes was varied from p_z/k=2, 3, 4 6 including aligned configuration at fixed p_x/k=3, where k is the roughness height. The results showed that the outer peak values of the Reynolds stresses are increased with increasing streamwise spacing, indicating the strong inner and outer-layer interaction at large p_x/k and the profiles of the Reynolds stresses with varying spanwise spacing show the similar trends in the outer region except p_z/k=4. In addition, we found that there is no difference between staggered and aligned cube geometries.

[1]This study was supported by the Creative Research Initiatives of NRF/MEST (No. 2011-0000423) of Korea.

3:54PM D7.00009 Direct Numerical Simulations of the flow over a dimpled flat plate , NIKOLAOS BERATLIS, KYLE SQUIRES, Arizona State University, ELIAS BALARAS, The George Washinton University — Golf balls use dimples to reduce the drag by as much as 50 percent when compared to a smooth sphere in the subcritical regime. Recent experiments and computations indicate that dimples introduce perturbations that drive high momentum fluid towards the wall, thus delaying separation and reducing drag. The nature of these perturbations and how they are affected by the Reynolds number, boundary layer thickness, and dimple shape is not well understood. In the present study we will report a series of DNS of the flow past a dimpled flat plate. A stability map with the important parameters that control the onset of the perturbations (Reynolds number, ratio of dimple depth to boundary layer thickness, etc) will be presented. We will show that dimples induce perturbations that lead to turbulent-like boundary layers at relatively low Reynolds numbers, where the flow would otherwise remain laminar. The origin of these perturbations is a thin shear layer forming over a dimple, which becomes unstable and generates vortices. These undergo complex three-dimensional instabilities transforming themselves into structures that resemble hairpin-like vortices typically found in turbulent wall bounded flows. They are very effective in mixing the flow and already within one dimple diameter downstream of the dimple the flow attains characteristics of low Reynolds number turbulence.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D11 Turbulence Modeling I 314

2:10PM D11.00001 Hybrid RANS/LES equations , MASSIMO GERMANO, Politecnico di Torino, MARTIN SANCHEZ-ROCHA, Dassault Systemes SIMULIA Corporation, SURESH MENON, Georgia Institute of Technology — The incompressible and compressible governing equations for the hybrid RANS/LES simulations have been recently formally derived by applying a hybrid filtering operator which linearly combines a LES average with the RANS statistical mean. These exact hybrid equations contain many additional terms that represent the interactions between RANS and LES formulations, and some preliminary simulations have shown their relevance in the correct representation of turbulence. Unfortunately the numerical implementation of these terms is not so easy, and a joint theoretical and numerical study is currently under work in order to overcome these difficulties and to develop simplified models for the hybrid contributions. New different procedures are explored and a new RANS-assisted LES that couples the additive hybrid procedure with an independent RANS computation is examined.

2:23PM D11.00002 Simulation of wall-bounded turbulent flow using ODTLES[1] , ALAN KERSTEIN, ESTEBAN GONZALEZ, RODNEY SCHMIDT, Sandia National Laboratories — Subgrid closure of coarse-grained simulations is challenging near walls and for multi-physics applications such as reacting flows. ODTLES [1] avoids coarse-grained advancement, yet is less costly than direct numerical simulation. In ODTLES, the smallest scales of motion are resolved on three arrays of sub-domains, such that each array fills the flow volume and provides full resolution in one coordinate direction. Within each sub-domain, one-dimensional turbulence (ODT) [2] simulates turbulent flow advancement in the resolved direction. A 3D advection step followed by pressure projection couples the sub-domains so as to capture large-scale 3D motion without requiring advancement of filtered flow variables. ODTLES has been used to simulate decaying homogeneous turbulence [1] and in the present study is extended to wall-bounded flows. The capability to capture 3D large-scale features while affordably resolving the wall-normal structure of near-wall flow is demonstrated.

[1] R. C. Schmidt, A. R. Kerstein, R. McDermott, Comput. Methods Appl. Mech. Eng. **199**, 865-880 (2010).
[2] A. R. Kerstein, Lect. Notes Phys. **756**, 291-333 (2009).

[1]Supported by the Division of Chemical Sciences, Geosciences, and Biosciences, Office of Basic Energy Sciences, U.S. Department of Energy.

2:36PM D11.00003 A New Formulation For Coupling Turbulence in Hybrid LES-RANS Techniques , STEPHEN WOODRUFF, Computational AeroSciences Branch, NASA Langley Research Center — In order for the full potential of hybrid Large-Eddy Simulation (LES) - Reynolds-Averaged Navier-Stokes (RANS) computations to be realized, the turbulence in the RANS and LES regions must be coupled, permitting RANS-LES transitions to be set according to the physics of the problem rather than the requirements of the method. A formulation for doing this is proposed which clarifies the relationship between quantities computed from blended models and physical variables when the numerical resolution is varied from that of RANS to that of LES. This understanding may then be employed to show how the governing equations themselves must be modified in order to maintain physical validity when resolution varies. The formulation thus permits accuracy to be maintained throughout LES-RANS transitions, regardless of where they are placed, and all turbulence dynamics are consequently coupled between LES and RANS subdomains. The effectiveness of this formulation is demonstrated in hybrid simulations of plane channel flow, with the LES-RANS transition placed in the log layer. The unphysical shift in the log layer is largely eliminated, at the cost of some additional computation. Prospects for further improvements in accuracy and efficiency are discussed.

2:49PM D11.00004 Wall-modeling in large-eddy simulation at high Reynolds number: an approach to predict accurate skin friction[1] , SOSHI KAWAI, The Institute of Space and Astronautical Science, JAXA, JOHAN LARSSON, Center for Turbulence Research, Stanford University — We present a new idea to address one of major sources of error in large-eddy simulation with wall-modeling where the wall shear stress is modeled directly: the inevitable presence of numerical and subgrid modeling errors in the first few grid points adjacent to the wall. By considering the behavior of turbulence length scales near the wall and the grid resolution, the cause of the error is diagnosed and a simple yet efficient approach to remove the impact of the error on the computed turbulence is proposed. The proposed approach allows us to feed accurate well-resolved information from the LES to the wall-model, thus allowing the wall-model to function as intended: this in turn leads to accurately predicted skin friction. The method is applied to zero-pressure-gradient attached and shock-induced separated supersonic turbulent boundary layers at very high Reynolds number, and compared with available experimental data.

[1]This work is supported in part by the NASA Fundamental Aeronautics Program - Hypersonics Project (Grant NNX08AB30A), and the JAXA International Top Young Fellowship.

3:02PM D11.00005 A wall model for large eddy simulation with complex geometries , JULIEN BODART, JOHAN LARSON, Center for Turbulence Research - Stanford University — Large eddy simulation of wall bounded flows is currently limited by the number of grid points required to resolve the inner part of the boundary layer at high Reynolds numbers. One possible solution is to compute approximate wall shear stresses through a separate-but-coupled RANS solver on a separate near-wall grid. This technique is implemented in an unstructured compressible solver. A fully structured grid is derived from the wall geometry to solve the RANS equations. The height of the RANS layer in relation to the boundary layer thickness is a critical parameter of the model and will be discussed. The capability of the model is assessed by computing several problems of both attached and separated flow at high Reynolds numbers, well beyond what is accessible to traditional LES.

3:15PM D11.00006 Experimental and Computational Investigations of Flow past Spinning Cylinders[1]

, PASQUALE CARLUCCI, IGBAL MEHMEDAGIC, LIAM BUCKLEY, DONALD CARLUCCI, U.S. Army ARDEC, Picatinny Arsennal, SIVA THANGAM, Stevens Institute of Technology — Experiments are performed in a low speed subsonic wind tunnel to analyze flow past spinning cylinders. The sting-mounted cylinders are oriented such that their axis of rotation is aligned with the mean flow. Data from spinning cylinders with both rear-mounted and fore-mounted stings are presented for a Reynolds numbers of up to 260000 and rotation numbers of up to 1.2 (based on cylinder diameter). Computations are performed using a two-equation turbulence model that is capable of capturing the effects of swirl and curvature. The model performance was validated with benchmark experimental flows and implemented for analyzing the flow configuration used in the experimental study. The results are analyzed and the predictive capability of the model is discussed.

[1]Funded in part by U. S. Army, ARDEC

3:28PM D11.00007 Evaluation of low Reynolds number turbulence models for open channel flow over a rough wall using high-resolution large eddy simulation (LES) data[1]

, SANDEEP BOMMINAYUNI, THORSTEN STOESSER, School of Civil and Environmental Engineering, Georgia Institute of Technology, Atlanta, GA, USA, NILS REIDAR OLSEN, Department of Hydraulic and Environmental Engineering, NTNU, Trondheim, Norway — Quite some effort has been put into the development of low Reynolds number turbulence models to enhance predictions of the near wall flow over smooth walls. Despite the success of these models analogous models for the flow over rough walls are sparse. Based on a high resolution large-eddy simulation of the flow over a bed artificially roughened by hemispheres, we test the applicability of several low Reynolds number models to rough wall flow. The Reynolds number of the flow based on the channel depth is Re=13,680 at a relatively low submergence of h/k=3.42, with h being the water depth and k the roughness height. Similar to flows over smooth walls, the near wall turbulent eddy viscosity requires some damping and damping functions developed for flows over smooth and rough walls are tested and evaluated. Moreover, in the log-layer i.e. at some distance away from the rough wall, the turbulent eddy viscosity is found to be smaller than over the smooth wall, suggesting damping over the entire water depth.

[1]Support for this study is received from NSF award 0738690.

3:41PM D11.00008 Uncertainty Quantification and Validation for RANS Turbulence Models

, TODD OLIVER, ROBERT MOSER, The University of Texas at Austin — Uncertainty quantification and validation procedures for RANS turbulence models are developed and applied. The procedures used here rely on a Bayesian view of probability. In particular, the uncertainty quantification methodology requires stochastic model development, model calibration, and model comparison, all of which are pursued using tools from Bayesian statistics. Model validation is also pursued in a probabilistic framework. The ideas and processes are demonstrated on a channel flow example. Specifically, a set of RANS models—including Baldwin-Lomax, Spalart-Allmaras, k-ϵ, k-ω, and $\overline{v^2}$-f—and uncertainty representations are analyzed using DNS data for fully-developed channel flow. Predictions of various quantities of interest and the validity (or invalidity) of the various models for making those predictions will be examined. This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

3:54PM D11.00009 A $k-\varepsilon$ Model for Flow through Submerged and Emergent Vegetation Accounting for Turbulence at the Stem Scale as Well as the Vertical Shear Scale(s)

, ALEXANDRA KING, EDWIN COWEN, Cornell University — Statistics of turbulence in and above vegetation are determined by the action and interaction of the turbulent wakes of plant stems (scaling with the stem diameter), the Kelvin-Helmholtz type vortices that form in regions of high vertical velocity gradient (scaling with the velocity gradient), and boundary layer turbulence (scaling with the flow depth). While turbulence from each of these sources is dissipated through the energy cascade, some shear-scale turbulence bypasses the lower wave numbers as shear-scale eddies do work against the form drag of the plant stems, converting shear-scale turbulence into wake-scale turbulence. We have developed a $k-\varepsilon$ model that accounts for all of these energy pathways. This is the first model of its kind to incorporate the stem diameter explicitly and thus predict the well-established relationship between stem diameter and turbulent kinetic energy in emergent vegetation. The model also performs well in submerged vegetation, where turbulent kinetic energy scales significantly, but to a lesser degree, with the stem diameter, and in real emergent aquatic vegetation, where multiple scales of vertical shear and stem diameter are equally important.

4:07PM D11.00010 The Oriented-Eddy Collision Model[1]

, MICHAEL B. MARTELL, J. BLAIR PEROT, University of Massachusetts Amherst — A novel method of treating turbulence - as a collection of interacting fluid particles (eddies) which have inherent orientation - is employed to capture fast pressure-strain in rapid distortion as well as other canonical turbulent flows. The Oriented-Eddy Collision (OEC) model is cast in the form of a collection of Reynolds-stress transport models. Underlying this approach is a unique PDF collision model. The model returns unsteady-RANS-like results, contains no special provisions to satisfy realizability, and maintains both frame and coordinate invariance. Simple wall-bounded flows, such as pressure driven channel flow, are captured without the use of wall-functions. Although more expensive than standard RST models, the model's accuracy and cost fall between those of RST and LES.

[1]This work was supported by the Office of Naval Research grant N00014-08-1-0275.

Sunday, November 20, 2011 2:10PM - 4:07PM –
Session D12 Turbulence Simulation II 315

2:10PM D12.00001 Scaling exponents of energy dissipation, enstrophy and scalar dissipation in high-resolution direct numerical simulations of turbulence[1]

, K.P. IYER, P.K. YEUNG, Georgia Tech, K.R. SREENIVASAN, New York Univ. — A central issue in the study of intermittency and refined similarity in turbulence is the behavior of higher-order moments of the locally-averaged energy dissipation rate (ϵ_r) within a domain of linear size r, especially for r in the inertial range. However, accurate determination of these quantities requires full-field knowledge of all components of velocity gradients which must be well-resolved in space, with the degree of difficulty increasing with the order of the moment. We examine a large DNS database in isotropic turbulence, with emphasis on well-resolved simulations up to 4096^3 at Taylor-scale Reynolds number 650. Efficient algorithms have been developed to extract the statistics of ϵ_r, and those of local averages of enstrophy and scalar dissipation rate, with averages taken both along a line of length r and within a cube of linear size r. Scaling exponents appearing in relations of the form $\langle\epsilon_r^p\rangle \propto r^{-\zeta_p}$ have been assessed, although the inference of a clear scaling range depends on numerical resolution in addition to Reynolds and Schmidt numbers. Data from 1D and 3D averages differ systematically, but both are useful for comparison with intermittency models in the literature.

[1]NSF Grants CBET-1139037 and OCI-0749223

2:23PM D12.00002 Dynamics of non-local interactions in isotropic turbulence , AGUSTIN MAQUI, DIEGO DONZIS, Texas A&M University — A large database of isotropic turbulence with R_λ ranging from 38 to 1100 and resolutions up to 4096^3 is used to study aspects of the dynamic response of the small scales to forcing at the largest scales. Time correlations of spectra and transfer show that changes in the large scales have an immediate effect on the smallest dissipative scales. Furthermore, these non-local interactions are strongly anti-correlated for wavenumbers beyond the so-called bottleneck. While the applied large-scale forcing is Gaussian, the probability density function of individual modes of the energy spectrum is skewed for all wavenumbers. On the other hand, transfer spectra shows departures from Gaussianity only at high wavenumbers. Short-term behavior is studied through the evolution of the ratio of spectral levels at different wavenumbers as forcing is abruptly introduced or discontinued. All results demonstrate the direct connection between distant scales. More importantly, the observed trends do not appear to decrease as the Reynolds numbers increases. Different models for the spectral transfer are shown to capture some of the observed behavior. Further consequences of the results will be discussed.

2:36PM D12.00003 Direct numerical simulation of top-down and bottom-up scalar diffusion in the convective Ekman layer[1] , SCOTT WAGGY, SEDAT BIRINGEN, Department of Aerospace Engineering Sciences, University of Colorado, Boulder — The turbulent Ekman layer commonly serves as a model of the atmospheric boundary layer. In this work we study the unstably-stratified turbulent Ekman layer by means of a direct numerical simulation. Studies have demonstrated that entrainment processes at the top of the atmospheric boundary layer affect turbulence within the mixed-layer. In order to differentiate between bottom-up and top-down diffusion, surface fluxes are separated from entrainment effects by monitoring passive scalars with conditions indicative of these processes. For each case, the scalar variance is parameterized as a function of z/z_i, the distance from the wall normalized by the temperature inversion height. The ability of these idealized variance functions to represent scalar flux through the convective boundary layer, where both bottom-up and top-down diffusion occur, is assessed.

[1]Support for Scott Waggy provided by the GAANN Fellowship. This research was supported by an allocation of advanced computing resources provided by the NSF. Computations were performed on Kraken at the NICS.

2:49PM D12.00004 Simulation of Homogeneous Turbulence Subjected to Plane Strain[1] , CHRIS ZUSI, J. BLAIR PEROT, University of Massachusetts, Amherst — Direct numerical simulation is used at a resolution of 512^3 to investigate the behavior of turbulence subjected to plane strain. The initial isotropic turbulence is generated by the stirring action of many small randomly placed cubes, rather than imposed as an initial condition. Anisotropic turbulent structure is then generated by dimensionless plane strain rates Sk/ϵ ranging from 2.2 to 54. Twenty different simulations are used to investigate the influence of initial conditions, strain rate, Reynolds number, and dimensionless strain time on the strained turbulence structure and its subsequent anisotropic decay.

[1]This work is support by the National Science Foundation.

3:02PM D12.00005 Studying Lagrangian dynamics of turbulence using on-demand fluid particle tracking in the JHU turbulence database[1] , HUIDAN YU, Mechanical Engineering, IUPUI, KALIN KANOV, ERIC PERLMAN, RANDAL BURNS, Computer Science, ALEXANDER SZALAY, Physics & Astronomy, GREGORY EYINK, Applied Mathematics & Statistics, CHARLES MENEVEAU, Mechanical Engineering, Johns Hopkins University — The JHU public turbulence database (http://turbulence.pha.jhu.edu) provides access to large datasets generated from DNS of turbulence, at present the output of a 1024^3 pseudo-spectral DNS of forced isotropic turbulence (Re_λ=443) with 1024 time-steps. The resulting 27 TB dataset can be accessed remotely through an interface based on the Web-services model allowing remote users to issue subroutine-like calls on their host computers. Here we describe the newly developed getPosition function: Given an initial position, integration time-step, as well as an initial and end time, the getPosition function tracks arrays of fluid particles inside the database and returns particle locations at the end of the trajectory integration time. GetPosition is applied to study Lagrangian velocity structure functions as well as tensor-based Lagrangian time correlation functions. The roles of pressure Hessian and viscous terms in the evolution of the strain-rate and rotation tensors are also explored.

[1]Supported by NSF (CDI-II, CMMI-0941530).

3:15PM D12.00006 On the Lagrangian Power Spectrum of Turbulence Energy in Isotropic Turbulence , FRANCESCO LUCCI, Aerothermochem. and Combustion Systems Laboratory, ETH, Zurich, Switzerland, VICTOR L'VOV, Dept. of Chem. Phys., The Weizmann Institute of Science, Rehovot 76100, Israel, ANTONINO FERRANTE, Dept. of Aeronautics & Astronautics, Univ. of Washington, Seattle, WA 98195, SAID ELGHOBASHI, Dept. of Mech. & Aerospace Engineering, Univ. of California, Irvine, CA 92697 — We present, for the first time, a derivation of the transport equation of the Lagrangian frequency power spectrum, $E_L(t,\omega)$, of turbulence energy in isotropic turbulence starting from the autocorrelation of the Lagrangian velocity. The new equation is: $\partial E_L(t,\omega)\big/\partial t = \mathcal{T}_L(t,\omega) - \varepsilon_L(t,\omega) + \Psi_L(t,\omega)$, where $\mathcal{T}_L(t,\omega)$ is the transfer rate of $E_L(t,\omega)$ across the frequency spectrum, $\varepsilon_L(t,\omega)$ is the viscous dissipation rate of $E_L(t,\omega)$, and $\Psi_L(t,\omega)$ is the external forcing rate. Our DNS shows that $\varepsilon_L(\omega)$ is maximum at low frequencies and vanishes at high frequencies. We also performed an analytical study which confirms the DNS result and shows that $\varepsilon_L(\omega) \sim (\omega_\eta - \omega)$, i.e. there is non-locality for $\varepsilon_L(\omega)$ in the ω domain, whereas $E_L(\omega) \sim (1/\omega^2 - 1/\omega_\eta^2)$, i.e. the locality is valid for $E_L(\omega)$, where ω_η is the Kolmogorov scale frequency.

3:28PM D12.00007 Dynamics of the intense vorticity structures near the turbulent/nonturbulent interface in jets , CARLOS DA SILVA, RICARDO REIS, IST - Technical University of Lisbon — Direct numerical simulations (DNS) of turbulent planar jets are used to analyse the dynamics of the intense vorticity structures (IVS) near the turbulent/nonturbulent (T/NT) interface in jets. Deep inside the jet shear layer the characteristics of the IVS are similar to the IVS found in many other flows: the mean radius, tangential velocity, and circulation Reynolds number are $R/\eta \approx 4.6$, $u_0/u' \approx 0.8$, and $Re_\Gamma/Re_\lambda^{1/2} \approx 28$, where u_0, and Re_λ are the root-mean-square of the velocity fluctuations and the Reynolds number based on the Taylor micro-scale, respectively. Statistics conditioned on the distance from the T/NT interface are used to analyse the effect of the T/NT interface on the dynamics of the IVS and show that the mean radius R, tangential velocity u_0 and circulation Γ of the IVS increases as the T/NT interface is approached, while the vorticity norm $|\omega|$ stays approximately constant. Unlike the IVS deep inside the shear layer, there is a small predominance of vortex diffusion over stretching for the IVS near the T/NT interface implying that the core of these structures is not stable $i.e.$ it will tend to grow in time.

3:41PM D12.00008 Effects of polymer additives on bulk turbulent energy transfer , ALEXANDRE DE CHAUMONT QUITRY, DOUGLAS H. KELLEY, NICHOLAS T. OUELLETTE, Yale University — We study the effect of long-chain polymer additives on fully developed bulk turbulence in an experimental von Karman swirling flow. Counterrotating impellers inject turbulent kinetic energy inertially in a cylindrical tank 0.6m wide, whose central region is studied using particle tracking. This experiment emphasizes bulk effects over drag reduction, which allows us to quantitatively compare energy dissipation in dilute polymer solutions with that of pure water. This work is supported by the National Science Foundation.

3:54PM D12.00009 Dynamics on the laminar-turbulent boundary and the origin of the maximum drag reduction asymptote in polymer solutions , MICHAEL GRAHAM, Univ. of Wisconsin-Madison, LI XI, Massachusetts Institute of Technology — Dynamical trajectories on the boundary in state space between laminar and turbulent plane channel flow – edge states – are computed for Newtonian and viscoelastic fluids. Viscoelasticity has a negligible effect on the properties of these solutions, despite the fact that their mean velocity profiles correspond closely to what is observed in experiments with drag-reducing polymer solutions in the maximum drag reduction regime. These results confirm the existence of weak turbulence states that cannot be suppressed by polymer additives, explaining the fact that there is an upper limit for polymer-induced drag reduction. The universality of this limit with regard to polymer properties arises from the fact that the edge states are too weakly three-dimensional to lead to persistent stretching of polymer chains.

Sunday, November 20, 2011 2:10PM - 4:07PM –
Session D13 Biofluids: Native and Prosthetic Cardiac Valves 316

2:10PM D13.00001 The Decay of Vortex Ring Circulation in Left Ventricular Filling , KELLEY STEWART, SUNGHWAN JUNG, Virginia Tech, WILLIAM LITTLE, Wake Forest University School of Medicine, PAVLOS VLACHOS, Virginia Tech — Radially confined vortex rings have been previously investigated and shown to exhibit a decay in circulation after vortex ring pinch-off. A semi-empirical model for the evolution of the vortex ring circulation subject to the effect of confinement was previously developed and displayed strong agreement with experimental observations. In this work the model was applied to clinical phase contrast Magnetic Resonance Imaging data to track and potentially predict the rate of vortex ring circulation decay within the filling left ventricle (LV) subject to changing physiological characteristics in normal and diseased conditions. From our previous work and clinical observations, we hypothesize that variations in vortex ring dynamics within the LV are caused by changes in ventricular geometry. Therefore impaired ventricular relaxation causes a decreased LV volume during filling, results in a more rapid decay of early diastolic hydrodynamic circulation. This model may be used to more completely understand the filling dynamics and potentially lead to improved diagnostic techniques.

2:23PM D13.00002 Measuring Heart Filling Propagation Velocity using the Cross Wavelet Transform , CASANDRA NIEBEL, Virginia Tech, TAKAHIRO OHARA, Wake Forest University Baptist Medical Center, PAVLOS VLACHOS, Virginia Tech, WILLIAM LITTLE, Wake Forest University Baptist Medical Center — During early diastole, a pressure gradient is formed across the mitral valve as the left ventricle (LV) relaxes, forcing blood from the left atrium into the LV. This process generates a rapid filling wave and creates an unsteady flow environment within the LV. A continuous wavelet transform is capable of dealing with non-stationary and noisy signals and is therefore ideal for measuring the wave speed of the early diastole rapid filling wave. This wave speed, or propagation velocity (Vp), is used clinically to evaluate diastolic function and is conventionally measured from a Color M-Mode (CMM) echocardiogram. A CMM scan gives a spatiotemporal map of the blood velocity in the left ventricle and is used to visualize flow patterns and manually measure the Vp. In this work, a moving cross wavelet transform is used to measure the phase shift between consecutive time steps in a CMM echocardiogram, providing a more robust and repeatable measurement of Vp, less sensitive to noise, aliasing boundaries, and user inputs.

2:36PM D13.00003 Experimental investigation of the flow field past a bileaflet mechanical heart valve in pulsatile flow within an anatomical aorta model , LAURA BROWN, STAVROS TAVOULARIS, University of Ottawa — A bileaflet mechanical heart valve (BMHV) has been mounted at the inlet of an anatomical model of the human aorta, and placed within a mock circulation loop that simulates physiological flow conditions. The working fluid matches the refractive index of silicone, from which the aorta model and other parts of the test section are made, and the viscosity of blood. Flow characteristics past the BMHV are measured using stereoscopic and planar particle image velocimetry and laser Doppler velocimetry. In contrast to previous experiments, in which heart valves have been tested in simplified aortic geometries, this arrangement permits the study of the dependence of flow past the valve upon recirculation in the sinuses of Valsalva, the flow rate through the coronary arteries, and the aorta curvature. The effect of valve orientation will also be investigated with the objective to determine a hemodynamically optimal configuration with potential benefits to implantation procedures. The measured viscous shear stress distribution will be analyzed towards predicting the initiation of thrombosis in patients and identifying regions of stagnation, which could facilitate thrombus attachment.

2:49PM D13.00004 Deformation Differences between Tricuspid and Bicuspid Aortic Valves in Vitro , KAI SZETO, UC San Diego, JAVIER RODRIGUEZ-RODRIGUEZ, UC3 Madrid, PETER PASTUSZKO, VISHAL NIGAM, JUAN C. LASHERAS, UC San Diego — It has been shown in clinical studies that patients with congenital bicuspid aortic valves (CBAVs) develop degenerative calcification of the leaflets at young ages compared to patients with the normal tricuspid aortic valves (TAVs). It has been hypothesized that the asymmetrical geometry of the leaflets in CBAVs, flow shear stresses (SS), disturbed flow, and excessive strain rate levels are possible causes for the early calcification and stenosis. Central to the validation of this hypothesis is the need to quantify the differences in strain rate levels between the BAVs and TAVs. We simulate the CBAVs by surgically stitching two of the leaflets of a porcine aortic valve together. To quantify strain differences, we performed in-vitro experiments in both trileaflet and bileaflet valves by tracking the motion of small ink dots marked on each leaflet surface. We then used phase-locked stereo photogrammetry to reconstruct at each instant of time the 3D surface of the leaflets and measure the strain rates in both radial and circumferential directions during the whole cardiac cycle. Our results indicate that the total strain rate of the simulated BAVs is about 15 to 20% higher than the normal leaflets of TAVs at systole. In the BAVs' case, the fused leaflet stretches radially up to 25% higher than the reference length. The excessive stretching in both directions in the fused leaflet results in large changes in the flow patterns and associated wall SS.

3:02PM D13.00005 Bicuspid aortic valve hemodynamics: a fluid-structure interaction study , SANTANU CHANDRA, CLARA SEAMAN, PHILIPPE SUCOSKY, University of Notre Dame — The bicuspid aortic valve (BAV) is a congenital defect in which the aortic valve forms with two leaflets instead of three. While calcific aortic valve disease (CAVD) also develops in the normal tricuspid aortic valve (TAV), its progression in the BAV is more rapid. Although studies have suggested a mechano-potential root for the disease, the native BAV hemodynamics remains largely unknown. This study aimed at characterizing BAV hemodynamics and quantifying the degree of wall-shear stress (WSS) abnormality on BAV leaflets. Fluid-structure interaction models validated with particle-image velocimetry were designed to predict the flow and leaflet dynamics in idealized TAV and BAV anatomies. Valvular function was quantified in terms of the effective orifice area. The regional leaflet WSS was characterized in terms of oscillatory shear index, temporal shear magnitude and temporal shear gradient. The predictions indicate the intrinsic degree of stenosis of the BAV anatomy, reveal drastic differences in shear stress magnitude and pulsatility on BAV and TAV leaflets and confirm the side- and site-specificity of the leaflet WSS. Given the ability of abnormal fluid shear stress to trigger valvular inflammation, these results support the existence of a mechano-etiology of CAVD in the BAV.

3:15PM D13.00006 The effect of mitral orifice eccentricity on the left ventricular hemodynamics[1] , TRUNG LE, FOTIS SOTIROPOULOS, University of Minnesota — We investigate the left ventricular hemodynamics using high resolution Direct Numerical Simulation. The LV geometry is reconstructed from Magnetic Resonance Imaging (MRI) data of a healthy volunteer. The diastolic kinematics of the LV wall is modelled using a cell-based electrical activation methodology, which yields global left-heart motion parameters well within the physiologic range of an adult. By prescribing the kinematics and the physiologic mitral valve waveform, numerical simulations are carried out to investigate the intraventricular flow patterns during the diastolic filling. The results show that the intraventricular flow is dominated by the formation and breakdown of a vortex ring originating from the mitral orifice. The eccentricity of the mitral orifice is found to be the determining factor controlling the dynamics of vortex formation and rotational flow patterns at the end of diastole.

[1]This work was supported by NIH Grant RO1-HL-07262 and the Minnesota Supercomputing Institute.

3:28PM D13.00007 Biofluid Dynamics in Cardiovascular System , HANSOL CHUNG, Vassar College, SU JUNG YOO, Cornell University, RICHARD KYUNG, Choice Research & Technology — Biofluid dynamics is characterized by the study of fluids in biological systems. Common biofluid systems include blood flow in the cardiovascular system and airflow in the lungs. The mathematical modeling of blood flow through the complex geometry of a prosthetic heart valve is a difficult task. In such a problem the complex geometries of the valve must be modeled properly so that they can be studied numerically. The present analysis is performed on a disk-type prosthetic heart valve. The valve is assumed to be in the aortic position and observed the structure of the valve cage influence the flow field near an aortic valve. For the purpose of mathematical modeling, the laminar incompressible two-dimensional steady flow of a homogeneous Newtonian fluid with constant viscosity is assumed. The flow is considered during the greater part of systole when the valve is fully open. Convergent numerical solutions are obtained for Reynolds numbers of 30, 180, 900 and 4500. Stream function, horizontal velocity, vertical velocity and shear stress solutions are computed at every grid point.

3:41PM D13.00008 Comparison of tricuspid and bicuspid aortic valve hemodynamics under steady flow conditions , CLARA SEAMAN, University of Notre Dame, JAMES WARD, University of Notre, PHILIPPE SUCOSKY, University of Notre Dame — The bicuspid aortic valve (BAV), a congenital valvular defect consisting of two leaflets instead of three, is associated with a high prevalence of calcific aortic valve disease (CAVD). CAVD also develops in the normal tricuspid aortic valve (TAV) but its progression in the BAV is more severe and rapid. Although hemodynamic abnormalities are increasingly considered potential pathogenic contributor, the native BAV hemodynamics remain largely unknown. Therefore, this study aims at comparing experimentally the hemodynamic environments in TAV and BAV anatomies. Particle-image velocimetry was used to characterize the flow downstream of a native TAV and a model BAV mounted in a left-heart simulator and subjected to three steady flow rates characterizing different phases of the cardiac cycle. While the TAV developed a jet aligned along the valve axis, the BAV was shown to develop a skewed systolic jet with skewness decreasing with increasing flow rate. Measurement of the transvalvular pressure revealed a valvular resistance up to 50% larger in the BAV than in the TAV. The increase in velocity between the TAV and BAV leads to an increase in shear stress downstream of the valve. This study reveals strong hemodynamic abnormalities in the BAV, which may contribute to CAVD pathogenesis.

3:54PM D13.00009 Deformation of a membrane in a pulsatile flow: implications in heart valve design , C. HERNANDEZ, J.E.V. GUZMAN, R. ZENIT, Universidad Nacional Autonoma de Mexico — Current designs of heart valves prosthetics have serious disadvantages and health issues for patients who use them. For this reason, a new design that combines durability (mechanical valves) and biocompatibility (biological valves) has to be conceived. Natural valves have very complex geometry because their leaflets have two principal curvatures, one imposed by the holding ring and a second one imposed by the bending of the closing arrangement. The objective of this research is to study the effects of both curvatures on the performance of a leaflet. It is well known that the increase of the curvature results in a larger stiffness, which, in turn, reduces the deflection of a leaflet. We conducted a study to determine the effect of changing the curvature (in two directions) of a flexible membrane when exposed to a steady and pulsatile flows. A study of the flow field that results from this interaction is also conducted by PIV measurements. Preliminary results of the leaflet deflection for many stiffnesses, curvatures and flow conditions will be presented and discussed.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D14 Bubbles I 317

2:10PM D14.00001 Inertial collapse and oscillations of a bubble in a compressible viscoelastic medium , ERIC JOHNSEN, University of Michigan, Ann Arbor, CHENGYUN HUA, California Institute of Technology — The inertial collapse and subsequent oscillations of a bubble in a compressible viscoelastic medium are studied theoretically and numerically in the context of therapeutic ultrasound. The focus of the present work is on the response of a bubble subjected to a step increase in pressure, i.e., Rayleigh collapse. Linear constitutive relations that include stress relaxation, elasticity, viscosity and strain rate relaxation are considered. A perturbation analysis is followed to estimate the damping, frequency and relaxation of the oscillations. The results are compared to numerical solutions of the Keller equation, showing good agreement over a wide range of parameters. The nonlinear coupling between viscosity, compressibility, elasticity and relaxation leads to unexpected bubble behavior, e.g., sustained oscillations when damping is expected. Direct simulations of the full three-dimensional equations of motion will be discussed, including viscous and viscoelastic effects, non-spherical behavior and nonlinear constitutive models.

2:23PM D14.00002 Modeling of a single clean bubble bouncing on a free surface , MASAO WATANABE, Hokkaido University, AYAKA SATO, MINORI SHIROTA, Hirosaki University, TOSHIYUKI SANADA, Shizuoka University — Bouncing of a rectilinearly rising bubble on a free surface is modeled with the use of a simple mass-spring system. We use an equation of motion of the system that consists of two springs connected in series, which allows us to account for the restoring forces of both the bubble and free surfaces, and a conservation equation of energy, which allows us to describe an exchange between the surface energy due to deformations of both the bubble and free surfaces and the kinetic energy of the bubble. We can determine that the contact time, i.e. the time duration of a bubble contacting a free surface, should be a half of the characteristic period of the oscillator. We also observe a single clean bubble bouncing on a flat free surface in ultrapure water in order to verify the present model. Analytical and experimental results agree quite well, even in the cases with significant surface deformations. When bubbles are smaller than 0.6 mm in radius, the deformations of both the bubble and free surfaces play important roles in bouncing. In the case of larger bubbles, bouncing is dominated by the free surface deformation since the bubble has already been highly deformed before collision.

2:36PM D14.00003 Bubble Motion in a Vibrating Liquid , T.J. O'HERN, B. SHELDEN, J.R. TORCZYNSKI, Sandia National Laboratories — Gas bubbles can be forced to move downward, overcoming the buoyancy force, by vertical vibration of the liquid containing the bubble. Bubble motion is controlled by interactions of the oscillating pressure field, and the corresponding bubble volume, with the drag force acting on the bubble. The bubble-drag asymmetry and the oscillating pressure gradient can combine to produce net bubble motion. This is analogous to the Bjerknes force in high-frequency vibrations. Experiments have been conducted demonstrating downward bubble motion over a range of vibration conditions (all less than 300 Hz), liquid properties, and pressure in the air above the free surface. Bubble size and velocity were measured using automated image-processing routines. Comparisons with theory and simulations will be shown. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

2:49PM D14.00004 Orbital revolution of a pair of bubbles in an acoustic field , MINORI SHIROTA, Hirosaki University, KOU YAMASHITA, TAKAO INAMURA — This experimental study aims to clarify the mechanism of orbital motion of two oscillating bubbles in an acoustic field. Trajectory of the orbital motion was observed using a high-speed video camera. Because of a good repeatability in volume oscillation of bubbles, we were also able to observe the radial motion driven at 24 kHz by stroboscopic like imaging; the cyclic bubble oscillation was appeared to slow down by capturing images at the framing rate close to the forcing frequency. The orbital motions of bubbles raging from 0.13 to 0.18 mm were examined with different forcing amplitude and in different viscous oils. As a result, we found that pairs of bubbles revolve along a circular orbit around the center of mass of the orbiting two bubbles. We also found that the two bubbles perform anti-phase radial oscillation. Although this radial oscillation should result in a repulsive secondary Bjerknes force, the bubbles kept a constant separate distance of about 1 mm, which indicates the existence of centripetal primary Bjerknes force. The angular velocity of orbital revolution increases linearly with the increase in Bjerknes force.

3:02PM D14.00005 On the influence of nanobubbles on the evaporation inception in liquids , DANIEL FUSTER, CNRS (UMR 7190), Universite Pierre et Marie Curie. Institut Jean le Rond d'Alembert , France, KIM PHAM, Eindhoven University of Technology Mechanical Engineering Materials Technology — We present a new approach for the calculation of the evaporation inception point in liquids. Using the derivative of the system's energy with respect to the void fraction as a stability criterion for the whole system, we consider that sudden evaporation only occurs when the bubble expansion is energetically favourable. The results obtained are shown to be equivalent to the Blake radius for a single spherical bubbles under tension. This method is proven to provide reasonable results also for superheated water. The method also allows gaining new insight into complex systems where nanobubbles stick to the walls. We provide new clues about future experimental measurements that should shed light into the process of bubble nucleation.

3:15PM D14.00006 Bubble motion and size variation during thermal migration with phase change , ASHA NURSE, GEOFFREY MCFADDEN, SAM CORIELL, NIST, MATHEMATICAL MODELING GROUP TEAM — An analysis of the motion of a spherical bubble in a two-phase, single component system with a vertical linear temperature gradient is presented. The model for the migration of an immiscible bubble considered by Young, Goldstein and Block is modified to allow for phase change at the bubble surface, including the possibility of both bubble translation and the change of bubble radius with time. Depending of the material parameters, the thermocapillary effects that normally lead to migration of an immiscible bubble can be overwhelmed by the effects of latent heat generation, resulting in a change in the mechanism driving the motion. For a water-steam system conditions are determined for a stationary bubble in which the the effects of buoyancy and thermal migration are balanced. The stability of the bubble is also considered.

3:28PM D14.00007 Film drainage of viscous liquids on top of bare bubble: Influence of the Bond number , FLORENCE ROUYER, LPMDI-UPEMLV, HELENA KOČÁRKOVÁ, SALAHEDINE METALLAOUI, FRANCK PIGEONNEAU, SVI-SGR, LPMDI- UNIVERSITÉ PARIS-EST MARNE LA VALLÉE TEAM, SVI- SAINT-GOBAIN RECHERCHE TEAM — We present experimental result of film drainage on top of gas bubbles pushed by gravity forces toward the upper surface of a liquid bath for Newtonian liquids with mobile interface (UCON, castor oil and soda-lime-silica melt). The temporal evolution of the thickness of the film between a single bubble and the air/liquid interface is investigated *via* interference method under various physical conditions, range of viscosities and surface tension of the liquids, and bubble sizes. These experiments evidence the influence of the deformation of the thin film on the thinning rate and confirm the slow down of film drainage with Bond number as previously reported by numerical work. A simple model that considered the liquid flow in the cap squeezed by buoyancy forces of the bubble is in good agreement with experimental and numerical data. Qualitatively, the smaller is the area of the thin film compare to the surface of the bubble, the faster is the drainage.

3:41PM D14.00008 Antibubble dynamics: slipping or viscous interfaces? , BENOIT SCHEID, Université Libre de Bruxelles, STÉPHANE DORBOLO, Université de Liège — An antibubble is a spherical thin air shell that is immersed in a surfactant mixture and drains under the action of hydrostatic pressure. Two lubrication models are proposed that account for either slipping or shear viscosity at the shell interfaces. Numerical solutions show that the antibubble lifetime decreases with increasing slip length or decreasing surface shear viscosity. Comparison with available experimental data is provided and gives some hints to discriminate between the slip and the surface viscous models, though a correspondence between these models is also established.

3:54PM D14.00009 Interaction and coalescence of two bubbles rising in a Hele-Shaw cell at high Reynolds numbers , PATRICIA ERN, SANDER HUISMAN[1], VERONIQUE ROIG, Institut de Mecanique des Fluides de Toulouse, France — We investigated the relative motion and deformation of two large bubbles released consecutively in a quiescent liquid confined in a thin-gap cell using a high-speed camera and image processing. Though the second bubble injected is smaller, we observed that in all cases it accelerates and catches up with the leading bubble. This acceleration is related to the wake of the leading bubble which also induces important changes in width and curvature of the trailing bubble. On the contrary, the velocity of the leading bubble is unaltered during the interaction. Shape adaptation of the two bubbles is observed prior to coalescence: the shape of the two bubbles together closely resembles a single large bubble. After the interfaces touch, the liquid film is drained at a constant velocity.

[1]Present Address: Physics of Fluids Group, University of Twente, NL

4:07PM D14.00010 An effect of zigzag and surface motions of a CO2 bubble on the mass transfer from the bubble to the surrounding liquid , MASAHIKO TORIU, Graduate School of Engineering, Shizuoka University, TAKAYUKI SAITO, Graduate School of Science and Technology, Shizuoka University — A bubbly flow is very useful for aeration, agitation and an enhancement of chemical reaction, etc. in industrial processes. In order to improve the efficiency of the industrial plants, we should acquire the knowledge of a bubbly flow: e.g. bubble motion, mass transfer and surrounding liquid motion. Using a single CO2 bubble well-controlled in its zigzag motion and surface oscillation, we discuss a relation between the bubble motions and the mass transfer. For this particular purpose, we precisely measured the bubble volume and the surface area, using two high-speed cameras and mirrors. By image processing, a 3D-bubble shape was reconstructed based on the projected bubble images. We regarded the volume/surface area of 3D-model as the volume/surface area of the examined bubble. In addition, instantaneous mass-transfer coefficients were calculated from these results. The mass transfer of the bubble was enhanced when the bubble trajectory shifted from the linear mode to the zigzag mode.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D15 Rayleigh-Benard Convection II: Instability and Oscillations 318

2:10PM D15.00001 Forecasting Flows from Target Pattern Instability in Rayleigh-Benard Convection , BALACHANDRA SURI, ADAM PERKINS, MICHAEL SCHATZ, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology — Using lab experiments combined with numerical simulations, we study systematically how the initial instability of an ordered pattern gradually evolves to a state of spatio-temporal complexity. The experiments begin from a reference pattern of axisymmetric convection rolls (a target pattern) that is reproducibly imposed using an optical technique for actuating fluid flow. For sufficiently large Rayleigh numbers, the axisymmetric pattern loses stability to patterns where the target's bull's-eye shifts off- center. We analyze an experimental ensemble of unstable patterns with nearby initial conditions to extract the spatial structure of the dominant modes and corresponding growth rates. We then test the extent to which a Boussinesq numerical model, in combination with a state estimation algorithm (Local Ensemble Transform Kalman Filter (LETKF)), can be used to predict the subsequent evolution of the experimentally observed patterns.

2:23PM D15.00002 Dynamic stabilization of the Rayleigh-Bénard instability in rectangular enclosures: A computational model[1] , RANDY CARBO, Pennsylvania State University, ROBERT SMITH, MATT POESE, Applied Research Laboratory, Pennsylvania State University — If an acoustic field is imposed on a fluid within a container, the critical Rayleigh number is a strong function of the frequency and amplitude of that acoustic field as noted by G. Swift. and S. Backhaus, [*J. Acoust. Soc. Am.* **126**(5), 2009]. Results will be reported for nonlinear and linear models constructed to predict the modified critical Rayleigh number, based on a full field solution of the hydrodynamic equations using the approach of A. Yu. Gelfgat, [*J. Comp. Phys.* **156**, 1999]. The spatial portion of the differential equations were solved using the Galerkin method and the dynamic stability for the linear model was determined using Floquet analysis. One of the benefits of the approach compared to the averaging methods used by G. Gershuni and D. Lyubimov, *Thermal Vibration Convection*, (Wiley, New York, 1998) is that the parametric stability boundary can be recovered. This study includes a variety of container aspect ratios, boundary conditions, and Rayleigh numbers ranging from 10^3 to 10^8.

[1]Research supported by the Office of Naval Research and ARL Exploratory and Foundational Research Program.

2:36PM D15.00003 Cellular Convection in a Chamber with a Warm Surface Raft , JOHN WHITEHEAD, Woods Hole Oceanographic Institution, ERIN SHEA, MIT, MARK BEHN, WHOI — We calculate velocity and temperature fields for Rayleigh-Benard convection in a chamber with a warm raft that can float along the top surface for Rayleigh number up to Ra=20,000. Two-dimensional, infinite Prandtl number, Boussinesq approximation equations are numerically advanced in time from a motionless state in a chamber of length L' and depth D'. We consider cases with an insulated raft and a raft of fixed temperature. Either oscillatory or stationary flow exists. The case of an insulated raft has three governing parameters: Ra, scaled chamber length L=L'/D', and scaled raft width W. For W=0 and L=1, the marginal state is at Ra=779.3. For smallest W (determined by numerical grid size) and Ra<790 the raft approaches the center monotonically in time. For 790<Ra<811 the raft has a decaying oscillation consisting of raft movement back and forth (and convection cell reversal). For 811>Ra>871 amplitude is steady, starting small and increasing with larger Ra and for Ra>871 raft movement ceases. For larger W, a range of W and Ra has raft oscillation up to Ra=20,000. Rafts in longer cavities (L=2 and 4) have almost no oscillatory behavior. With a raft of temperature Tr rather than insulating, Ra=20,000, and with internal heating, there are wider ranges of oscillating flow. Thus the presence or absence of motion is very sensitive to W, L, raft thermal properties and Ra. Reasons why are discussed.

2:49PM D15.00004 Spiral Defect Chaos in Generalized Swift-Hohenberg Models with Mean Flow[1] , ALIREZA KARIMI, Virginia Tech, ZHI-FENG HUANG, Wayne State University, MARK PAUL, Virginia Tech — We present a numerical study of spiral defect chaos in two generalized models of the Swift-Hohenberg equation with mean flow. We use large-sized square domains with periodic boundaries and integrate the equations for very long times (up to one million time units) to study the effect of mean flow on pattern dynamics as the strength of mean flow is varied. The magnitude of mean flow is adjusted via a continuous parameter characterizing the type of the velocity boundary conditions on the horizontal surfaces in a convective flow. We show that the typical parameters used in the literature for spiral defect chaos studies yield a weak mean flow magnitude and leads to a state dominated by large and slowly moving target defects. When the magnitude of mean flow is sufficiently large, it is demonstrated that spatiotemporal chaos is feasible as indicated by a positive Lyapunov exponent. In addition, we compare the spatial distribution of the mean flow for these models with that of large aspect ratio Rayleigh-Bénard convection undergoing spiral defect chaos and discuss their differences near spiral defects.

[1]NSF Award CBET-0747727, Virginia Tech's Advanced Research Computing Center.

3:02PM D15.00005 1:2 Resonance and Pattern Formation in Thermal Convections , KAORU FUJIMURA, Tottori University — Resonant interaction of steady modes having wavenumbers in the ratio 1:2 was re-examined on a hexagonal lattice in two-layered Rayleigh-Benard convection, where the exact resonance may take place between critical modes. Amplitude equations were derived by means of the center manifold reduction. Numerical examination of the equations reveals that the heteroclinic orbit of AGH-type exists, but loses its stability with respect to disturbances in four-dimensional complex space. Instead, a new type of heteroclinic cycle arises in rather narrow range of parameters. The cycle is a connection between rolls and hexagons.

3:15PM D15.00006 Natural Convection Due to a Long Wavelength Heating , ALI ASGARIAN, MOHAMMED Z. HOSSAIN, JERZY M. FLORYAN, University of Western Ontario — Natural convection in an infinite horizontal layer subject to a spatially periodic heating is considered. The analysis is focused on the heating wave number $\alpha \rightarrow 0$. It has been shown that convection has a simple topology consisting of one pair of counter-rotating rolls per heating period when the heating intensity does not exceed the critical value of the Rayleigh number Ra = 427. Secondary motions in the form of rolls aligned in the direction of the primary rolls and concentrated around the hot spots occur for more intense heating. When 427<Ra<470 the secondary motions are described by supercritical pitchfork bifurcations and can occur only if α is reduced below 0.14. One of the branches of such bifurcations is associated with an odd number of secondary rolls per half period, with rolls at the hot spots rotating in the direction opposite to the primary rolls. The other branch is associated with an even number of secondary rolls per half period, with the rolls at the hot spots co-rotating with the primary rolls. The number of rolls increases without limit as α decreases with new rolls being pinched off in pairs. Increase of heating intensity to Ra>470 results in secondary motions occurring at larger values of α, i.e., α >0.14, and bifurcation changing character into "bifurcations from infinity." It is shown that the observed phenomena are strictly associated with the small wave number limit of the external heating.

3:28PM D15.00007 Natural Convection in a Fluid Layer Subject to Periodic Heating , MOHAMMED Z. HOSSAIN, JERZY M. FLORYAN, University of Western Ontario — Natural convection in a horizontal layer exposed to periodic heating is considered. The primary response leads to stationary convection in the form of rolls orthogonal to the heating wave vector. For large α a uniform conductive layer emerges at the upper section of the fluid layer. Secondary convection gives rise either to the longitudinal rolls, or to the transverse rolls or to the oblique rolls depending on α. Three mechanisms of instability have been identified. In the case of small and moderate α the parametric resonance leads to the pattern of instability that is locked-in with the pattern of the heating as $\delta_{cr} = \alpha/2$, where δ_{cr} denotes component of the disturbance wave vector. The second mechanism is associated with the formation of patterns of vertical temperature gradients and the primary convection currents, operates approximately in the same range of α and provides direct modulation with structure dictated by α. The third mechanism operates for large α where the instability is driven by the uniform mean vertical temperature gradient created by the primary convection with the critical disturbance wave vector $\delta_{cr} \rightarrow$ 1.56. Competition between the first and second effects gives rise to the appearance of soliton lattices.

3:41PM D15.00008 Flow simulation for a high Prandtl number liquid heated from above[1] , ERDEM UGUZ, Univ. Florida, Department of Chemical Eng., FL 32611, FRANCK PIGEONNEAU, Surface du Verre Interfaces, CNRS/St. Gobain, France, GERARD LABROSSE, Univ. Paris-Sud 11, France, RANGA NARAYANAN, Univ. Florida, Department of Chemical Eng., FL 32611, UNIVERSITY OF FLORIDA TEAM, SAINT-GOBAIN TEAM, UNIVERSITY OF PARIS-SUD 11 TEAM — Natural convection in a glass furnace has crucial importance for glass manufacturing as it affects mixing and bubble dynamics. To understand the convective flows in a glass furnace where the Prandtl numbers typically are greater than 1000, a numerical study is performed using the spectral collocation Chebyshev method. Calculations were done for a 2D rectangular geometry for various aspect ratios (i.e. height divided by length) and Rayleigh (Ra) numbers for a constant viscosity fluid. Depending upon the value of Ra, and for a fixed aspect ratio, secondary and even tertiary transient cell formations were observed. In the process of heating (i.e. in the process of increasing Ra) thermal and velocity boundary layers are formed. Also with increasing Ra the solution becomes unstable and the critical Ra number that defines this transition has a strong dependence on the aspect ratio.

[1]Support from NSF OISE 0968313, the Partner University Fund and a Chateaubriand fellowship are acknowledged.

3:54PM D15.00009 Relaxation Dynamics of Spatiotemporal Chaos in the Nematic Liquid Crystal[1] , FAHRUDIN NUGROHO, TATSUHIRO UEKI, YOSHIKI HIDAKA, SHOICHI KAI, Department of Applied Quantum Physics and Nuclear Engineering, Graduate School of Engineering, Kyushu University, Fukuoka 819-0395, Japan — We are working on the electroconvection of nematic liquid crystals, in which a kind of spatiotemporal chaos called as a soft-mode turbulence (SMT) is observed. The SMT is caused by the nonlinear interaction between the convective modes and the Nambu–Goldstone (NG) modes. By applying an external magnetic field **H**, the NG mode is suppressed and an ordered pattern can be observed. By removing the suppression effect the ordered state relax to its original SMT pattern. We revealed two types of instability govern the relaxation process: the zigzag instability and the free rotation of wavevector **q**(**r**).

[1]This work is partially supported by Grant-in-Aid for Scientific Research (Nos. 20111003, 21340110, and 21540391) from the Ministry of Education, Culture, Sport, Science, and Technology of Japan and the Japan Society for the Promotion of Science (JSPS)

4:07PM D15.00010 Rayleigh-Bénard-Poiseuille flow: Optimal growth of streamwise-uniform disturbances , J JOHN SOUNDAR JEROME, JEAN-MARC CHOMAZ, PATRICK HUERRE, Ecole Polytechnique — An investigation of the dominant transient growth mechanisms in plane Poiseuille flow subjected to a destabilizing cross-stream temperature gradient is presented. It was pointed out by the same authors in DFD meeting 2009 that only the streamwise-uniform and nearly-streamwise-uniform disturbances are highly influenced by the Rayleigh number **Ra** and Prandtl number **Pr**. Here, it is demonstrated that the *short-time* behavior is governed by the classical inviscid lift-up mechanism and the optimal input for the largest *long-time* response is given by the adjoint of the dominant eigenmode with respect to the energy scalar product: the Rayleigh-Bénard eigenmode without its streamwise velocity component. These short and long-time responses are then shown to depict, up to leading order, the optimal transient growth **G(t)**. It is thereby brought out that, at moderately large **Ra** (or small **Pr** at a fixed **Ra**), the dominant adjoint mode is a good approximation to the optimal initial condition for all time. The results remain qualitatively similar over a general class of norms that can be considered as growth functions. For instance, the dominant adjoint eigenmode still approximates the maximum optimal response.

Sunday, November 20, 2011 2:10PM - 4:20PM – Session D16 Experimental Techniques II 319

2:10PM D16.00001 Tomographic particle image velocimetry over a triangular prism in unidirectional flows , IN MEI SOU, National Research Council Postdoctoral Fellow, JOSEPH CALANTONI, Marine Geosciences Division, Naval Research Laboratory, Code 7434 — Using tomographic particle image velocimetry (Tomo-PIV), the full three-dimensional-three-component (3D-3C) flow structure and turbulence characteristics over a triangular prism in a recirculating water tunnel were investigated. Here we present preliminary results from a new Tomo-PIV system for subcritical Froude number flows. Large-scale vortex shedding from the tip of the triangular prism is observed. Results of coherent structure organization analyzed by 3D vorticity calculation will be presented. Using the full 3D-3C instantaneous velocity field, turbulent kinetic energy is directly evaluated without any of the assumptions often needed for 2D PIV measurements. Details of the experimental setup including a unique device designed to perform our Tomo-PIV volume calibration will be discussed. We perform an in-depth turbulent kinetic energy budget and explore the feasibility of extending the measurement technique to other complex flows.

2:23PM D16.00002 Simultaneous 3D Strain and Flow Fields Measurement of a Model Artery under Unsteady Flows[1] , MOSTAFA TOLOUI, JIAN SHENG, University of Minnesota, AEM — Fluid-Structure Interaction imposes challenges in both aero-elasticity and biomedical studies. A simultaneous solid deformation and fluid flow measurement technique based on digital in-line holographic particle tracking velocimetry (PTV) has been developed. It allows us to measure concurrently 3D strain field of a deforming structure and the unsteady flow near it. To facilitate the measurement, both wall and flow are seeded with tracer particles distinguished by size. The motion of these tracers provides the 3D deformation of the wall and the 3D velocity distribution of the flow separately. A fully index matched facility including transparent artery and NaI solution is constructed to enable observations near the wall or through the complex geometry. An arterial model with the inner diameter of 9.5 mm and the thickness of 0.9 mm is manufactured from the cross-linked transparent PDMS at the mixing ratio of 1:10 and doped with mono-dispersed 19 μm polystyrene particles. A cinematic holographic PTV system is used to trace the 3D particle motion in the model and flow simultaneously. Preliminary study is performed within a sample volume of $15 \times 15 \times 75$ mm with the spatial resolution of 7.4 μm in lateral and 10 μm in depth. Uncertainty and accuracy analysis will be reported.

[1]NSF Grant No: CBET-0844647

2:36PM D16.00003 Simultaneous 3D measurement of flow velocities and an arbitrarily moving surface[1] , YOUNG JIN JEON, HYUNG JIN SUNG, KAIST — Simultaneous 3D measurement technique for flow velocities and an arbitrarily moving surface has been developed by applying image processing methods to the tomographic PIV. Particles and a surface were captured simultaneously through a conventional PIV system without additional cameras or an illumination device. To adjust the optimal exposure times of particles and the surface in the similar time scales, red fluorescent tracer particles were used and long pass glass filters were installed in the lens system. The volume self calibration was employed for improving the 3D reconstruction qualities. Particles and a surface image could be separated by using an image separation. 3D surface shapes were then reconstructed by applying the feature detection and the morphological evaluation to the surface image. Subsequently, particle volumes were reconstructed by MLOS-SMART. The surface reconstruction procedure was validated for various known shaped bodies. Experiments for flows around a rotating cylinder, a flag motion and a flapping flag were carried out to investigate the applicability of the present technique.

[1]This study was supported by the Creative Research Initiatives of NRF/MEST (No. 2011-0000423) of Korea.

2:49PM D16.00004 Visual hull method for tomographic PIV of flow around moving objects[1] , DEEPAK ADHIKARI, ELLEN LONGMIRE, University of Minnesota — Measurement of velocity around arbitrarily moving objects is of interest in many applications. This includes flow around marine animals and flying insects, flow around supercavitating projectiles, and flow around discrete drops or particles in multiphase flows. We present a visual hull technique that employs existing tomographic PIV reconstruction software to automate identification, masking and tracking of discrete objects within a three-dimensional volume, while allowing computation and avoiding contamination of the surrounding three-component fluid velocity vectors. The technique is demonstrated by considering flow around falling objects of different shape, namely a sphere, cube, tetrahedron and cylinder. Four high-speed cameras and a laser are used to acquire images of these objects falling within liquid seeded with tracer particles. The acquired image sets are then processed to reconstruct both the object and the surrounding tracer particles. The reconstructed object is used to estimate the object location at each time step and mask the reconstructed particle volume, while the reconstructed tracer particles are cross-correlated with subsequent particle volumes to obtain the fluid velocity vectors.

[1]Supported by NSF IDBR Grant #0852875.

3:02PM D16.00005 Recent advancements in tracer particle based-interface reconstruction , MASSIMILIANO ROSSI, CHRISTIAN CIERPKA, RODRIGO SEGURA, CHRISTIAN J. KÄHLER — Tracer particles are commonly used in experimental fluid dynamics to probe the velocity of a flow. An example are particle tracking velocimetry methods (PTV), in which the velocity field is obtained by tracking the position of individual tracer particles in a control volume at different time instants. It has been recently demonstrated how this approach can be extended to successfully reconstruct the interface between two fluid phases. This procedure has some advantages compared to traditional methods using dye visualization, since the diffusion coefficient of tracer particles is considerably smaller than the one of molecular dyes and the 3D topology of an interface can be obtained from a single recording (using 3D-PTV), with no need of scanning procedures. The main limitation of this approach is clearly due to the resolution of the reconstruction that is constrained to the tracer particle density. In this work, the latest developments of this method are presented, in particular showing the possibility to perform 3D-time-resolved measurements. Results on the shape of droplets and free-surface waves are provided.

3:15PM D16.00006 Volume Segmentation and Ghost Particles[1] , ISAAC ZISKIN, RONALD ADRIAN, Arizona State University — Volume Segmentation Tomographic PIV (VS-TPIV) is a type of tomographic PIV in which images of particles in a relatively thick volume are segmented into images on a set of much thinner volumes that may be approximated as planes, as in 2D planar PIV. The planes of images can be analysed by standard mono-PIV, and the volume of flow vectors can be recreated by assembling the planes of vectors. The interrogation process is similar to a Holographic PIV analysis, except that the planes of image data are extracted from two-dimensional camera images of the volume of particles instead of three-dimensional holographic images. Like the tomographic PIV method using the MART algorithm, Volume Segmentation requires at least two cameras and works best with three or four. Unlike the MART method, Volume Segmentation does not require reconstruction of individual particle images one pixel at a time and it does not require an iterative process, so it operates much faster. As in all tomographic reconstruction strategies, ambiguities known as ghost particles are produced in the segmentation process. The effect of these ghost particles on the PIV measurement is discussed.

[1]This research was supported by Contract 79419-001-09, Los Alamos National Laboratory

3:28PM D16.00007 Iterative Reconstruction of Volumetric Particle Distribution for 3D Velocimetry , BERNHARD WIENEKE, LaVision GmbH, DOUGLAS NEAL, LaVision, Inc — A number of different volumetric flow measurement techniques exist for following the motion of illuminated particles. For experiments that have lower seeding densities, 3D-PTV uses recorded images from typically 3-4 cameras and then tracks the individual particles in space and time. This technique is effective in flows that have lower seeding densities. For flows that have a higher seeding density, tomographic PIV uses a tomographic reconstruction algorithm (e.g. MART) to reconstruct voxel intensities of the recorded volume followed by the cross-correlation of subvolumes to provide the instantaneous 3D vector fields on a regular grid. A new hybrid algorithm is presented which iteratively reconstructs the 3D-particle distribution directly using particles with certain imaging properties instead of voxels as base functions. It is shown with synthetic data that this method is capable of reconstructing densely seeded flows up to 0.05 particles per pixel (ppp) with the same or higher accuracy than 3D-PTV and tomographic PIV. Finally, this new method is validated using experimental data on a turbulent jet.

3:41PM D16.00008 Analysis of an Optimized MLOS Tomographic Reconstruction Algorithm and Comparison to the MART Reconstruction Algorithm , RODERICK LA FOY, PAVLOS VLACHOS, Virginia Tech — An optimally designed MLOS tomographic reconstruction algorithm for use in 3D PIV and PTV applications is analyzed. Using a set of optimized reconstruction parameters, the reconstructions produced by the MLOS algorithm are shown to be comparable to reconstructions produced by the MART algorithm for a range of camera geometries, camera numbers, and particle seeding densities. The resultant velocity field error calculated using PIV and PTV algorithms is further minimized by applying both pre and post processing to the reconstructed data sets.

3:54PM D16.00009 Quantitative visualization of fine-scale three-dimensional flow structures in turbulence using time-resolved stereoscopic scanning Particle Image Velocimetry , YE CHENG, FRANCISCO JAVIER DIEZ — A time-resolved stereoscopic scanning particle image velocimetry (TR-SSPIV) system is developed to investigate the fine-scale 3D structures in free shear turbulent jets. The system provides a simultaneous measurement of the three-component velocity field in a three-dimensional volume (3D3C) with Kolmogorov-scale (η) resolution, offering a true representation of the complete nine-component velocity gradient tensor. Quantitative visualization of the coherent structures at fine-scale turbulence is obtained and four basic structural shapes (sheets, tube, square ribbons and spherical blobs) are identified as building blocks of complex turbulent structures. The local acceleration $\partial \mathbf{u}/\partial t$ is obtained and represented as 3D structures, which showed a strong anti-alignment with the convective acceleration terms. A novel vortex identification scheme is introduced based on the local pressure. This method gives more direct description of vortex cores, compared to previously published ones including enstrophy, Q, λ_2 and Δ criteria. Extensive statistical analyses are performed to study the probability density function (PDF), joint PDF, and spectra of the velocity gradients, enstrophy production rate and energy dissipation rate to compare with isotropy theory.

4:07PM D16.00010 Three-dimensional measurement of the laminar flow field inside a static mixer , MICHEL SPEETJENS, RENE JILISEN, PAUL BLOEMEN, Eindhoven University of Technology — Static mixers are widely used in industry for laminar mixing of viscous fluids as e.g. polymers and food stuffs. Moreover, given the similarities in flow regime, static mixers often serve as model for compact mixers for process intensification and even for micro-mixers. This practical relevance has motivated a host of studies on the mixing characteristics of static mixers and their small-scale counterparts. However, these studies are primarily theoretical and numerical. Experimental studies, in contrast, are relatively rare and typically restricted to local 2D flow characteristics or integral quantities (pressure drop, residence-time distributions). The current study concerns 3D measurements on the laminar flow field inside a static mixer using 3D Particle-Tracking Velocimetry (3D-PTV) Key challenges to the 3D-PTV image-processing procedure are the optical distortion and degradation of the particle imagery due to light refraction and reflection caused by the cylindrical boundary and the internal elements. Ways to tackle these challenges are discussed and first successful 3D measurements in an actual industrial static mixer are presented.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D17 Taylor Couette Instability 320

2:10PM D17.00001 Axial velocimetry and torque scaling in turbulent Taylor-Couette flow with independently rotating cylinders , HANSEN NORDSIEK, University of Maryland at College Park, MATTHEW PAOLETTI, University of Texas at Austin, DANIEL LATHROP, University of Maryland at College Park — We present experimental studies investigating axial flow velocities and torque scaling in the turbulent flow of water between two independently rotating cylinders. The Taylor-Couette system is capable of both strong turbulence ($Re > 2 \times 10^6$) and rapid rotation. The axial velocity profile near one end (via ultrasound doppler velocimetry), the torque required to rotate the inner cylinder, and the wall shear stress at one point on the outer cylinder are precisely measured as a function of the angular velocities of the two cylinders. We compare our measurements with previous experiments and discuss the potential relevance to angular momentum transport in astrophysical flows.

2:23PM D17.00002 Exchange of Stabilities in Couette Flow between Cylinders with Navier-Slip Conditions , PABLO SUAREZ, Delaware State University, ISOM HERRON, Rensselaer Polytechnic Institute — Viscous Couette flow is derived for flow between two infinitely long concentric rotating cylinders with Navier slip on both. Its axisymmetric linear stability is studied within a regime that would be hydrodynamically stable according to Rayleigh's criterion: opposing gradients of angular velocity and specific angular momentum, based on the rotation rates and radii of the cylinders. Stability conditions are analyzed, by methods based on those of Synge and Chandrasekhar. For sufficiently small slip length on the outer cylinder no instability occurs with arbitrary slip length on the inner cylinder. As a corollary, slip on the inner cylinder is shown to be stabilizing, with no slip on the outer cylinder. Two slip configurations are investigated numerically, first with slip only on the outer cylinder, then second with equal slip on both cylinders. It is found that instability does occur (for large outer slip length), and the principle of exchange of stabilities emerges. The instability disappears for sufficiently large slip length in the second case; Rayleigh's criterion provides an explanation for these phenomena.

2:36PM D17.00003 Wavy toroidal and helicoidal vortices in Taylor-Couette with axial and radial through-flows[1] , DENIS MARTINAND, Aix-Marseille Université, ERIC SERRE, CNRS, NILS TILTON, University of Maryland, RICHARD LUEPTOW, Northwestern University — Based on previous stability analyses of a Taylor-Couette cell with superimposed axial and/or radial through-flows, a weakly non-linear approach is used to determine the saturated states of the toroidal or helicoidal marginal instabilities above their relevant thresholds. These saturated states are then used to compute secondary instabilities in the form of wavy vortices by Floquet analysis. Different scenarios for this secondary transition are assessed, involving harmonic or subharmonic, convective or absolute, secondary modes. The effects of the axial and radial through-flows on these secondary modes and their critical conditions are then addressed. The analytical results are compared to direct numerical simulations of supercritical flows using spectral methods.

[1]Supported by Agence Nationale de la Recherche ANR-08-BLAN-0184-03.

2:49PM D17.00004 Stability and angular momentum transport of flows between co-rotating cylinders , MARC AVILA, MPI Dynamics and Selforganization — The flow of gas in astrophysical disks is characterized by radially decreasing angular velocity and radially increasing angular momentum. In principle, disk-like flows can be obtained in experiments of fluid between co-rotating cylinders and so these are often used to extrapolate momentum transport and stability of accretion disks. Recent results from experimental setups in Princeton and Maryland reach, however, opposite conclusions as whether keplerian flows remain laminar or become turbulent at high Reynolds numbers. In this contribution direct numerical simulations of flows with the precise geometry and boundary conditions of the Princeton and Maryland experiments will be reported. It is found that in both cases the flows become turbulent at moderate Reynolds numbers. The underlying instabilities are due to axial boundary conditions in the experiments.

3:02PM D17.00005 Linear Analysis of Oscillations in Free Surface Taylor-Couette Flows[1] , M. PRETKO, Princeton University, E.M. EDLUND, E. SPENCE, A.H. ROACH, H. JI, Princeton Plasma Physics Laboratory — Determining the mechanism of angular momentum transport in accretion disks is a long-standing problem in astrophysics. In addition to the magnetorotational instability, another possible mechanism is hydrodynamic turbulence. Previous experimental work made use of a bounded Taylor-Couette system and found no transition to turbulence even at Reynolds numbers of order 10^6. However, a free surface adds another degree of freedom and allows for modes which are absent in the bounded case. Therefore, a free surface may lead to instabilities which could not be observed in previous experiments. As preparations for experiments with a free surface Taylor-Couette device are made, we perform a linear analysis of oscillations in such a system and search for unstable modes. As a first approximation, surface tension is neglected and effects of viscosity are dealt with as small corrections. The experimental apparatus has a split endcap in order to minimize Ekman circulation. Thus, in this analysis, it is assumed that the background flow is strictly azimuthal. Analytic and numerical work is currently in progress and will ultimately be compared with experiment.

[1]This work was supported by PPPL's National Undergraduate Fellowship Program.

3:15PM D17.00006 Recent results from the Princeton MRI experiment , ERIK SPENCE, Princeton Plasma Physics Laboratory, AUSTIN ROACH, ERIC EDLUND, Princeton Plasma Physics Laboratory, CHRISTOPHE GISSINGER, Princeton University, PETER SLOBODA, HANTAO JI, Princeton Plasma Physics Laboratory — The magnetorotational instability (MRI) is widely expected to be responsible for the observationally inferred accretion rates of astrophysical disks. The Princeton MRI experiment is a Taylor-Couette device used to generate conditions under which the MRI should be unstable: an externally applied magnetic field, and a radially decreasing angular velocity profile. The apparatus' unique feature is independently-rotating endcap rings, which are used to reduce Ekman circulation. The working fluid is GaInSn; its velocity field is measured using an ultrasonic Doppler velocimetry system. Though an ideal-Couette angular rotation profile can almost be attained, through careful choice of end-cap ring speeds, residual Ekman circulation remains. The instability's identification is complicated by this secondary circulation, since it moves in the same radial direction as the flow expected from the instability. Comparison of radial flows in MRI-stable and MRI-unstable regimes is used to search for the instability's signature, as determined by numerical simulations. An update on the search for the instability will be presented.

3:28PM D17.00007 Observation of a magnetically enabled instability in the Princeton MRI Experiment , A.H. ROACH, E.J. SPENCE, C. GISSINGER, E.M. EDLUND, P. SLOBODA, H. JI, Princeton Plasma Physics Laboratory, Princeton University — The Princeton MRI Experiment is a modified Taylor-Couette device with a GaInSn working fluid used for the study of rotating MHD flows. An Ultrasound Doppler Velocimetry (UDV) system is used to measure the velocity field. It has revealed an instability causing large-amplitude velocity fluctuations when an axial magnetic field is applied to both hydrodynamically stable and hydrodynamically unstable background flow states with the split axial endcaps rotating differentially. The azimuthal velocity has a characteristic spiral mode structure at saturation, with an azimuthal mode number m=1. This instability appears in a region of parameter space distinct from that where the magnetorotational instability is expected to be present. Nonlinear 3D simulations have shown an instability of the Shercliff layer that forms at the split endcaps when a magnetic field is applied, and the resultant azimuthal flow patterns are largely consistent with experimental observations. Work is ongoing to measure the Shercliff layer in the experiment, and to identify the precise mechanism for the instability in the simulations. Results from experiments and simulations will be presented. Supported by DOE contract DE-AC02-09CH11466.

3:41PM D17.00008 Study of Magnetorotational Instability in a Swirling Plasma

, HANTAO JI, KYLE KREMER, ERIC EDLUND, ERIK SPENCE, Princeton Plasma Physics Laboratory, Princeton U. — Fast angular momentum transport in accretion disks has been an outstanding problem in astrophysics for more than three decades. The magnetorotational instability (MRI) has been identified as a powerful mechanism to transport angular momentum. Experiments using liquid metal are underway to study the MRI in incompressible MHD limit. A new frontier in accretion disk research is to explore physics beyond incompressible MHD. Possible new effects include compressibility, multiple-fluid effects, kinetic effects, ion-neutral collisions, radiation pressure, and dust grains. A swirling gas flow with quasi-Keplerian profiles, which are characterized by radially increasing angular momentum with decreasing angular velocity, is set up by an injection-pumping system. Spiral antennas are used to transmit RF power into the experiment through the helicon mode of discharge to ionize the gas with a desirable degree of ionization. A wide range of outstanding issues can be studied in such device, including: nonlinear hydrodynamic instability, baroclinic instability with axial or azimuthal temperature gradient, MRI in weakly ionized plasmas with Hall effect and ambipolar diffusion. Theoretical analyses and experimental explorations will be presented.

3:54PM D17.00009 Searching for a subcritical transition in quasi-Keplerian flows

, E.M. EDLUND, Princeton Plasma Physics Laboratory, M. PRETKO, Princeton University, A. ROACH, P. SLOBODA, E.J. SPENCE, H. JI, Princeton Plasma Physics Laboratory — Angular momentum transport in hot accretion disks in binary star systems and in active galactic nuclei is likely governed by the presence of the magneto-rotational instability (MRI) which can be effective in the presence of very weak magnetic fields. Colder, proto-planetary accretion disks with negligible ionization may not be able to support the MRI and consequently, a hydrodynamic path to turbulence may be needed to enhance the frictional forces and the transport of angular momentum. Hydrodynamic experiments in Taylor-Couette devices with controlled boundary conditions have shown that quasi-Keplerian flows are quiescent with very low levels of fluctuations. Yet there remains the possibility that these prior studies either have not accessed a nonlinear or subcritical transition to turbulence. We report here on recent studies in the Hydrodynamic Turbulence eXperiment (HTX), an order unity aspect ratio Taylor- Couette device, at the Princeton Plasma Physics Laboratory where quasi-Keplerian flows at Reynolds numbers of order 10^6 are probed with active perturbations to search for a subcritical transition. The role and regulation of secondary circulation in these experiments will be discussed.

4:07PM D17.00010 Tomographic PIV Observations of Turbulent Structures in Transitional Taylor-Couette Flow[1]

, DANIEL BORRERO, MICHAEL SCHATZ, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology — Theoretical and numerical studies have suggested that unstable, exact solutions of the Navier-Stokes equations known as Exact Coherent Structures (ECS) may provide a foundation for a simplified dynamical description of turbulence. Taylor-Couette flow (TCF) is an ideal system to make experimental connections to current ECS theory since it maintains some of its assumptions (streamwise periodic boundary conditions and plane Couette flow (in the small-gap limit)), but also includes realistic effects (no-slip spanwise boundary conditions). Furthermore, when only the outer cylinder is allowed to rotate, the system exhibits a subcritical transition to turbulence much like the one that the systems used to develop ECS theory undergo. We have developed a Taylor-Couette system with a series of jets built into the inner cylinder wall, which we can use to make small, local perturbations to the flow. We use tomographic particle image velocimetry to measure the structures that form when the flow is perturbed and how these structures affect the stability of the laminar flow in the transitional regime.

[1]Supported by NSF grant CBET-0853691.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D18 Microfluids: General II 321

2:10PM D18.00001 Programming fluid flow with microstructures

, HAMED AMINI, MAHDOKHT MASAELI, DINO DI CARLO, UCLA — Flow control and fluid interface manipulation in microfluidic platforms are of great importance in a variety of applications. Current approaches to manipulate fluids generally rely on complex designs, difficult-to-fabricate 3D platforms or use of active methods. Here we show that in the presence of simple cylindrical obstacles (i.e. pillars) in a microchannel, at moderate to high flow rates, streamlines tend to turn and stretch in a manner that, unlike intuition for Stokes flow, does not precisely reverse after passing the pillar. The asymmetric flow behavior up- and down-stream of the pillar due to fluid inertia manifests itself as a total deformation of the topology of streamlines that effectively creates a *net* secondary flow which resembles the recirculating Dean flow in curving channels. Confocal images were taken to investigate the secondary flow for a variety of microstructure settings. We also developed a numerical technique to map the fluid motion in the channel which is utilized to characterize the secondary flow as well as to engineer the fluid patterns within the channel. This passive method creates the possibility of exceptional control of the 3D structure of the fluid within a microfluidic platform which can significantly advance applications requiring fluid interface control (e.g. optofluidics), ultrafast mixing and solution control around cells.

2:23PM D18.00002 Mapping low Reynolds number cavity flow phenomena inside microfluidic devices

, RAMY FISHLER, JOSUE SZNITMAN, Technion- Israel institute of technology — Small-scale cavity flows are known for their usefulness in microfluidic applications. These include devices such as passive micro-mixers and cell diagnostic applications. Concurrently, small-scale cavity flows are physiologically relevant in capturing respiratory flow structures pertinent to the alveolar region of the lung. However, studies in this latter field are typically restricted to computational fluid dynamic (CFD) simulations and scaled-up experimental models based on hydrodynamic similarity matching. In the present study, we investigate low Reynolds number cavity flow phenomena using a microfluidic screening platform featuring rectangular channels with cylindrical cavities. Based on experiments and CFD simulations, we map the detailed cavity flow patterns characterizing a wide range of dimensionless geometric parameters and Reynolds numbers. We find that attached flow is observed at low aspect ratios and large opening angles, while separated flow, characteristic of alveolar flows, is associated with high aspect ratios and small opening angles. These findings provide design guidelines for microfluidic cavity applications and serve as a steppingstone towards designing in vitro microfluidic models of alveolated airways.

2:36PM D18.00003 Distribution of two miscible fluids at a T-junction

, CASEY KARST, BRIAN STOREY, JOHN GEDDES, Olin College — When a fluid comprised of multiple phases or constituents flows through a network, how the phases distribute throughout the network is a fundamental and important question. If one focuses at a single bifurcation where an incoming branch splits into two daughter branches, many systems have a phase fraction in the daughter branches which is different than the incoming branch. In this work, laminar stratified flow of two miscible fluids with different viscosity at a T junction is explored with experiments and simulation. The phase distribution as a function of flow ratio in the daughter branches is explored. The distribution depends on the incoming Reynolds number, viscosity ratio, and incoming volume fraction. Good agreement between experiment and simulation is found. The results may be relevant for microfluidic networks with two or more fluids, or networks involving blood flow (either microfluidic or microvascular) where the red blood cells and plasma distribute unevenly throughout the network.

2:49PM D18.00004 Flow distribution in 4 channels junctions at low Reynolds numbers as a function of the junction angle[1] , J.P. HULIN, L. TALON, D. ETIEN, H. AURADOU, UPMC and Paris-Sud Universities, FAST Lab., Orsay (France)., M. CACHILE, GMP-FIUBA, Buenos-Aires (Argentina), J.M. GOMBA, IFAS, UNCPBA, Tandil (Argentina) — Flow distribution in junctions is a key issue in microfluidics processes. We study a junction between 4 straight channels facing each other by pairs and in which fluid is injected in two facing channels at a low Reynolds number (fluorescent dye is added in one channel). LIF measurements are performed in transparent millimetric models together with 2D FEM simulations using the Stokes equation. For equal injection flow rates on both sides, the fraction of the injected fluorescent fluid leaving in the outflow channel at the lowest angle α from the injection is always larger than 0.5 (or equal for $\alpha = 90°$) and increases as α decreases. Surprisingly, this fraction becomes equal to 1 below a threshold non-zero value $\alpha_c \simeq 30°$ (the fluid "bounces back" at the junction). For $\alpha \leq \alpha_c$, recirculation cells appear at the center of the junction and increase in size and number as α goes to zero. Moving the LIF measurement plane perpendicular to that of the junction shows a $2D$ structure of the flow at low Reynolds numbers while $3D$ features appear at larger ones.

[1]Research supported by the LIA : Physics and Mechanics of Fluids.

3:02PM D18.00005 μPIV characterization of a toroidal microfluidic vortex driven by opto-electrokinetic methods , JAE-SUNG KWON, STEVE WERELEY, Purdue University — The simultaneous application of a uniform AC electric field and a focused laser to a fluid induces a toroidal microvortex with its center at the laser beam waist. In this paper we analyze the vortex quantitatively using μPIV technique. Also the vortex is characterized with respect to electric field strength, AC frequency, laser power and nonlinear coupling effect of the thermal gradient in the fluid and the electric field. As a result the flow vorticity at a certain constant AC frequency and laser power is found to increase as the square of the electric field strength. And it does not change appreciably in 10–70 kHz range under a fixed electrical voltage and laser power, but starts decreasing from the frequency ($\sim$100 kHz) at which a charge relaxation of the fluid occurs. Also at a constant AC frequency and voltage, the vorticity is significantly enhanced as the laser power is increased from 20 to 170mW due to the nonlinear interaction of the two driving sources—the laser and electric field. These results provide important insights to optimize the design and operation of a novel microvortex based mixer for rapid, dynamic on-chip mixing in progresses.

3:15PM D18.00006 Light field particle image velocimetry , BRYCE MCEWEN, Brigham Young University, JESSE BELDEN, Naval Undersea Warfare Center, TADD TRUSCOTT, Brigham Young University — Three-dimensional flow field measurement of microscopic environments requires innovative solutions. We propose a system capable of measuring instantaneous, three- dimensional velocities in microscopic flow fields using light field microscopy. The light field microscope used in these experiments consists of a camera that images the back focal plane of a micro lens array (similar to the system proposed by Levoy 2006). The lens array enables capture of the light field in a single image, which can then be reparameterized to render synthetically-refocused images at different focal depths post-capture. A three-dimensional volume can be reconstructed from this synthetic focal stack, and particles extracted for velocity measurements.

3:28PM D18.00007 Measurements of turbulent velocity statistics in a microscale rectangular confined impinging jets reactor , MICHAEL OLSEN, VISHWANATH SOMASHEKAR, RODNEY FOX, Iowa State University — Microscale chemical reactors capable of operating in the turbulent flow regime, such as the confined impinging jets reactor (CIJR), offer many advantages for rapid chemical processing at the microscale, especially in application such as flash nanoprecipitation used for the production of functional nanoparticles. In the presented work, microscopic particle image velocimetry (microPIV) was employed on a microscale rectangular CIJR to obtain instantaneous velocity fields at jet Reynolds numbers of 200, 1000 and 1500, which corresponds to completely laminar, weakly turbulent, and fully turbulent regimes respectively in the reaction zone. For each Reynolds number, approximately 2000 instantaneous velocity fields were collected to analyze the flow fields and calculate pointwise and spatial turbulence statistics. Large eddy simulation (LES) was then performed to obtain time resolved simulated velocity fields which were then compared with the experimental results. Good agreement was observed between the experimental results and the LES results, demonstrating the viability of LES could be used as a tool for designing microscale reactors.

3:41PM D18.00008 Some turbulent like flow property observed in microfluidics , G. WANG, F. YANG, W. ZHAO, University of South Carolina — We report some flow properties that are similar to what normally observed in many conventional turbulent flows, can also be achieved in an electrokinetic flow in microfluidics where the Reynolds number (Re) is in the order of 0.1. One important issue in microfluidic devices is the relatively slow mixing due to laminar flow at low Reynolds number. In conventional fluid dynamics, mixing can be enhanced by manipulate flow into turbulent flow. However, in mirofluidics, where Re is usually < 1, it is always believe that the flow is laminar. When mixing is enhanced (e.g. using AC electrical excitation) in microfluidics, it is believed that the flow is chaotic, not turbulence. If there is turbulence in microfluidics, another challenge is how to measure it. Here AC electrokinetics is used to excite a pressure driven flow in a microchannel with conductive sidewall to generate turbulent like flow at Re in the order of 0.1. First the flow is found to be random. With a recently developed laser induced fluorescence photobleaching anemometer (LIFPA) having high temporal and spatial resolution, we observe quantitatively random fluctuation of flow velocity and concentration in the flow. The scalar mixing is so rapid that the flow exhibits diffusion property in conventional turbulent flows. Visualization indicate that the flow may not be cahotic. Furthermore, we also observe that the power spectra of velocity and concentration exhibited a continuous decay with a span of more than one decade, again, indicating a multiscale property of turbulent flow.

3:54PM D18.00009 Application of interferometry in analyzing gas flow in microchannels[1] , YONGLI LI, DAVID NEWPORT, University of Limerick, SHIJU JOSEPH, Unversity of Limerick, JUERGEN J. BRANDNER, Karlsruhe Institute of Technology — Interferometry is a noninvasive measurement and based on this technique, the measurement or visualization of changes in physical properties of transparent objects can be achieved by detecting the refractive index changes. Gas pressure and temperature can be related to their refractive index, so interferometry can be used for local measurement of changes of these properties along the channel. A Mach–Zehnder interferometer was built with a laser with a wavelength of 633 nm, a high speed camera and two acousto-optic modulators (AOMs). Due to small channel characteristic length and sometimes low gas pressure, the measurement could be much influenced by noise. The AOMs can introduce frequency shifts into the system by acousto-optic effect, which can avoid mechanical noise generally by translating piezoelectric mirrors. The channel sides with optical access are made from crystalline silica (Quartz) that does not show speckle effects and any laser absorbance. For initial tests, the local gas pressure drop distribution along microchannel is studied at room temperature.

[1]The research leading to these results has received funding from the European Community's Seventh Framework Programme (ITN - FP7/2007-2013) under grant agreement no. 215504.

4:07PM D18.00010 Ultrasonic cleaning of root canals , BRAM VERHAAGEN, CHRISTOS BOUTSIOUKIS, University of Twente, The Netherlands, LEI-MENG JIANG, RICARDO MACEDO, Academic Center for Dentistry Amsterdam, The Netherlands, LUC VAN DER SLUIS, Paul Sabatier University, Toulouse, France, MICHEL VERSLUIS, University of Twente, The Netherlands — A crucial step during a dental root canal treatment is irrigation, where an antimicrobial fluid is injected into the root canal system to eradicate all bacteria. Agitation of the fluid using an ultrasonically vibrating miniature file has shown significant improvement in cleaning efficacy over conventional syringe irrigation. However, the physical mechanisms underlying the cleaning process, being acoustic streaming, cavitation or chemical activity, and combinations thereof, are not fully understood. High-speed imaging allows us to visualize the flow pattern and cavitation in a root canal model at microscopic scales, at timescales relevant to the cleaning processes (microseconds). MicroPIV measurements of the induced acoustic streaming are coupled to the oscillation characteristics of the file as simulated numerically and measured with a laser vibrometer. The results give new insight into the role of acoustic streaming and the importance of the confinement for the cleaning of root canals.

Sunday, November 20, 2011 2:10PM - 4:07PM –
Session D19 Flow Control I 322

2:10PM D19.00001 Validated model of arc-filament plasma actuators for control of wall-bounded flows[1] , DANIEL BODONY, MAHESH NATARAJAN, University of Illinois at Urbana-Champaign — Plasma actuators based on the electrical arcs between two electrodes have shown promise in controlling high-subsonic and low-supersonic flows. Simulation-based predictions of these flows have often used heuristic models for the effect the plasma has on the flow to be controlled. In this talk we present a two-parameter model of the actuator which combines the unsteady Joule heating induced by the plasma with a thermally perfect model of air. PIV and spectroscopy data are used, in conjunction with simulations, to understand the two parameters and demonstrate how their values are to be determined. The importance of the cavity in which the electrodes are mounted is discussed, as is the role of diffusion. We demonstrate the use of the actuator model by controlling a high-subsonic, separating boundary layer in an S-duct geometry.

[1]Supported by the Rolls-Royce Corporation

2:23PM D19.00002 Numerical Study for Separation Control Mechanism of Impulse Actuation[1] , SOLKEUN JEE, NASA Ames, OMAR LOPEZ MEJIA, Universidad de los Andes, Colombia, ROBERT MOSER, University of Texas at Austin — A flow control mechanism by which impulse actuation delays flow separation is investigated numerically. The actuation produces a short-duration high-velocity jet, which exploits the sensitivity of separated flow to momentary actuation. Previous experimental and numerical studies have shown that this actuation disrupt the separated region on a stalled airfoil, reattaching the boundary layer. This actuation, which is spatially as well as temporally localized, globally alters the baseline flow over long time (100 times the actuation time). The computations reported here provide detailed flow structure associated with the actuation and the separated flow. The flow modification includes four major stages following an impulse actuation: disruption of the separated region, vorticity extraction from the boundary layer, the reattachment and return to stall. It was hypothesized that the disruption of the separating layer is resulted from interactions with the vortices produced by the actuation. This was tested by artificially introducing similar vortices up- and down-stream of the nominal separation. Results are consistent with the hypothesis and show that a complete disruption of the separated shear layer is required for the desired flow modifications.

[1]Supported by AFOSR and ORAU

2:36PM D19.00003 Axisymmetric Synthetic Jets: A Momentum-based Modeling Approach , XI XIA, Department of Mechanical and Aerospace Engineering, University of Florida, KAMRAN MOHSENI, Department of Mechanical and Aerospace Engineering and Department of Electrical and Computer Engineering, University of Florida — A dimensional analysis approach is presented in order to offer a unified modeling approch for axisymmetric continuous and synthetic jets. The synthetic jet is injected into a quiescent air environment and the flow field is measured in the far field using hot-wire anemometry. Four parameters are identified to control the far field of the jet. These are: the spatial position, jet width, centerline velocity, and momentum flux. The functional dependency of these parameters is given by dimensional analysis. Experimental data show that in the far field, the spreading rate and decay rate of the jet are constants and the momentum flux is also conserved. Therefore, two nondimensional parameters are proposed to represent the characteristics of the far field of a synthetic jet. Furthermore, the relationship between the two constants are given theoretically and verified experimentally in this study. The results of several continuous jets are also presented for comparison. The validity of this work provides universal model for the far field of both synthetic jets and continuous jets regardless of the configurations of jet actuators or the activation conditions. It can be also concluded from this model that the momentum flux and the virtual eddy viscosity are actually the key parameters controling the characteristics of a synthetic jet far field.

2:49PM D19.00004 ABSTRACT WITHDRAWN –

3:02PM D19.00005 3D Numerical Analysis of Flow Control on Wind Turbine Blades , ONKAR SAHNI, Rensselaer Polytechnic Institute, ERTAN KARAISMAIL, CCNI/SCOREC, Rensselaer Polytechnic Institute — Wind turbine blades are exposed to unsteady and spatially-varying loadings in a real field. These loadings result in fluctuating structural forces which in turn lead to failure of blades as well as gearbox. In this study, we perform numerical analysis of flow over a wind turbine blade placed in a wind tunnel; where dynamic motions are imposed to the blade in order to emulate scenarios observed in a real field. Furthermore, we also study the effect of active flow control (via synthetic-jets) on unsteady aerodynamic characteristics of the blade under dynamic motions; the idea is to be able to control aerodynamic loads and mitigate failures. Numerical analysis is based on massively parallel simulations using hybrid turbulence models. Comparisons with experimental data will also be included.

3:15PM D19.00006 Aerodynamic Performance Enhancement of a Finite Span Wind Turbine Blade using Synthetic Jets , KEITH TAYLOR, CHIA MIN LEONG, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Modern wind turbines undergo significant changes in pitch angle and structural loading through a revolution. Recent developments in flow control techniques, coupled with increased interest in green energy technologies, have led to interest in applying these techniques to wind turbines, in an effort to increase power output and reduce structural stress associated with widely varying loading. This reduction in structural stress could lead to reduced operational costs associated with the maintenance cycle. The effect of active flow control on the aerodynamic and structural aspects of finite span blade was investigated experimentally. When synthetic jets were employed the effect on aerodynamic performance and structural vibrations, during static and dynamic pitch conditions, was significant. In order to investigate if the jets can be actuated for less time (reduce their power consumption), they were actuated during only a portion of the pitch cycle or using pulse modulation. The results showed that these techniques result in significant reduction in the hysteresis loop and the structural vibrations.

3:28PM D19.00007 Numerical simulation of tandem-cylinder noise-reduction using plasma-based flow control[1] , MENG WANG, AHMED ELTAWEEL, FLINT THOMAS, ALEXEY KOZLOV, University of Notre Dame, DONGJOO KIM, Kumoh National Institute of Technology, Korea — The noise of low-Mach-number flow over tandem cylinders at $Re_D = 22,000$ and its reduction using plasma actuators are simulated numerically to confirm and extend earlier experimental results. The numerical approach is based on large-eddy simulation for the turbulent flow field, a semi-empirical plasma actuation model, and Lighthill's theory for acoustic calculation. Excellent agreement between LES and experimental results is obtained for both the baseline flow and flow with plasma control in terms of wake velocity profiles, turbulence intensity, and frequency spectra of pressure fluctuations on the downstream cylinder. The validated flow-field results allow an accurate acoustic analysis based on Lighthill's equation, which is solved using a boundary-element method. The effectiveness of plasma actuators for reducing noise is demonstrated. In the baseline flow, the acoustic field is dominated by the interaction of the downstream cylinder with the upstream wake. With flow control the interaction noise is reduced drastically through suppression of vortex shedding from the upstream cylinder, and the vortex-shedding noise from the downstream cylinder becomes dominant. The peak sound pressure level is reduced by approximately 15 dB.

[1]Supported by NASA Cooperative Agreement NNX07AO09A

3:41PM D19.00008 Identification of frequency lock-on using Koopman spectral analysis[1] ,

JONATHAN TU, CLARENCE ROWLEY, Princeton University, SHAWN ARAM, RAJAT MITTAL, Johns Hopkins University — The separated flow past an airfoil at a high angle-of-attack is characterized by the frequencies of the wake, shear layer, separation bubble, and zero-net-mass-flux (ZNMF) actuation (if applied). In certain configurations, "lock-on" may occur, in which some or all of these frequencies take on the same value. Previous studies have shown that the presence of lock-on may be related to the effectiveness of ZNMF forcing at a particular frequency. As a model for separated airfoil flows, we analyze the high Reynolds number flow past a finite-thickness flat plate with an elliptical leading edge, inducing separation via a blowing and suction boundary condition rather than angle-of-attack. 3-D large eddy simulations are performed and the resulting data are analyzed using Koopman spectral analysis and proper orthogonal decomposition (POD). Koopman analysis clearly identifies the characteristic flow frequencies and provides corresponding spatial modes. From the spatial support in these modes we determine whether or not lock-on occurs and which structures (wake, shear layer, or separation bubble) are involved. A combination of Koopman and POD analyses shows that ZNMF forcing is most effective when lock-on is achieved in the most energetic modes.

[1]Supported by AFOSR grant FA9550-09-1-0257, NSF GRFP.

3:54PM D19.00009 On the interaction of von Karman shedding and sinusoidal actuation ,

KUNIHIKO TAIRA, Florida A&M University/Florida State University — Modifying the dynamics of unsteady separated flow around a bluff body is of engineering importance, as it directly relates to lift and drag on the body. We numerically investigate how the use of sinusoidal flow control input (mimicking synthetic jet actuator) alters the laminar wake dynamics dominated by the von Karman vortex shedding behind a circular cylinder. This study considers the influence of actuator location, forcing amplitude, and forcing frequency, with particular focus on lock-on. Also analyzed is the resulting change in the force experienced by the cylinder to extract useful operating condition for potential applications in feedback control and vibration suppression.

Sunday, November 20, 2011 2:10PM - 4:20PM – Session D20 Interfacial/Thin Film Instability II 323

2:10PM D20.00001 Influence of Shear Stress Jumps in Self-Assembly and Growth of Nanopillars by Thermocapillary Forces ,

SANDRA TROIAN, NAN LIU, California Institute of Technology, 1200 E. California Blvd. MC 128-95, Pasadena, CA — Recent theoretical and experimental studies have confirmed how thermocapillary forces can spontaneously generate periodic, 3D nanopillar arrays in viscous nanofilms exposed to a very large and uniform transverse thermal gradient [1,2]. These formations are quite sensitive to local growth conditions and exhibit variations in location, in-plane symmetry and growth rates. Here we explore the role of shear stress jumps in localizing and self-assembling pillar arrays for more rapid and regular formations. Finite element simulations and Fourier analysis of growing waveforms are used to quantify trends observed when a molten viscous nanofilm is held adjacent to a cold protrusion with sharp sidewalls. The sidewalls enforce a jump in the lateral thermal gradient along the free surface of the film which triggers formation of nanopillar trains. We investigate their number, height, spacing and growth rate as a function of the magnitude in shear stress jump and initial noise amplitude. We discuss why the characteristic wavenumber closely matches predictions from linear stability theory despite the large amplitude perturbation.

[1] M. Dietzel and S. M. Troian, Phys. Rev. Lett. 103, 074501 (2009); J. Appl. Phys. 108, 074308 (2010)
[2] E. McLeod, Y. Liu, and S. M. Troian, Phys. Rev. Lett. 106, 175501(2011)

2:23PM D20.00002 The Electrohydrodynamics of Superimposed Fluids Subjected to Nonuniform Electric Fields ,

ASGHAR ESMAEELI, Southern Illinois University Carbondale — The interaction of electric field with fluid interfaces has been a problem of long-standing interest; however, there is currently a resurgent of interest on the subject because of its potential applications in a host of novel microfluidic processes. A particularly attractive application is the electrolithography, where increasingly small structures can be formed in a polymeric liquid by application of a "uniform" electric field on a pair of patterned electrodes that enclose the liquid. Motivated by this and similar examples, here, we study the behavior of the interface separating two leaky dielectric fluids that are confined between two horizontal flat plates and are subjected to a nonuniform (sinusoidal) electric field. We perform Direct Numerical Simulation (DNS) using a front tracking/finite difference scheme and solve the governing electrohydrodynamics equations in the framework of Taylor's leaky dielectric theory. The equilibrium shapes of the interface are determined as a function of the controlling parameters of the problem such as the relative thicknesses of the two fluids.

2:36PM D20.00003 Thermocapillary Control of Thin Fluid Sheets ,

BURT TILLEY, WPI, MARK BOWEN, Waseda University — We consider the evolution and rupture dynamics of a thin viscous planar sheet subject to a symmetric initial long-wave disturbances in the thermal and velocity fields. We apply a long-wave analysis in the limit where deviations from the mean sheet velocity are small, but thermocapillary stresses, fluid inertia, van der Waals effects, capillarity, and heat transfer to the environment can be significant. From a linear stability analysis, we find that a stable thermal mode couples the velocity and interfacial dynamics. The phase difference between the initial temperature and velocity distributions is a key parameter in determining the time to rupture. Using phase modulation in the initial temperature along with amplitude modulation in the initial velocity allows the potential for control of the rupture patterns along the sheet. Self-similar rupture dynamics in the limit of zero van der Waals forces are also presented.

2:49PM D20.00004 Electrosprays generated by a DC electric field[1] ,

DEMETRIOS PAPAGEORGIOU, DEVIN CONROY, RICHARD CRASTER, Imperial College London, HSUEH-CHIA CHANG, Notre Dame University, OMAR MATAR, Imperial College London — The breakup of an electrified jet in a gas with an axially applied electric field is investigated theoretically. The jet fluid is taken to be a symmetric electrolyte and proper modelling of the cationic and anionic species is used by considering the Nernst-Planck equations in order to find the volume charge density that influences the electric field in the jet. The governing equations are investigated asymptotically in the long wave limit and the one dimensional model is solved numerically as a function of the hydrodynamic, electrical, and electrokinetic parameters. The electric field causes the jet to stretch and thin to a point where ion repulsion forces the jet to undergo Rayleigh fission. We measure the distance at which this point occurs by comparing the jet radius to the distance at which ion repulsion is important.

[1]Royal Academy of Engineering and EPSRC (EP/E046029/1)

3:02PM D20.00005 Breakup of compound viscous threads with electrostatic and electrokinetic effects[1] , RICHARD CRASTER, DEVIN CONROY, DEMETRIOS PAPAGEORGIOU, OMAR MATAR, Imperial College London — The breakup of electrified viscous compound jets surrounded by a dielectric gas are investigated theoretically. The fluids are considered to be electrolytes and the core fluid viscosity is assumed to be much larger than that of the annular fluid. Axisymmetric configurations are considered with the three fluids bound by a cylindrical electrode that is held at a constant voltage potential. The model equations are investigated asymptotically in the long wave limit, yielding two cases corresponding to a negligible surface charge with electrokinetic effects and a leaky dielectric model. A linear stability analysis for both cases is performed and the electrical effects are found to have a stabilizing effect, which is consistent with previous investigations of single electrified jet breakup at small wave numbers. The one-dimensional equations are also solved numerically. The electric field is found to cause satellite formation in the core fluid, which does not occur in the purely hydrodynamic case, with the satellite size increasing with the strength of the electric field.

[1]EPSRC Grant number EP/E046029/1

3:15PM D20.00006 Spreading, retraction, and sustained oscillations of surfactant-laden lenses[1] , OMAR MATAR, GEORGE KARAPETSAS, RICHARD CRASTER, Imperial College London — We examine the dynamics of surfactant laden-lenses spreading over liquid substrates. We use the lubrication theory in combination with models for the spreading process, the effects of surfactant at the contact line and the sorption kinetics above and below the critical micelle concentration. Our model is solved numerically using a finite-element formulation. We carry out a full parametric study and simulate a wide range interesting behaviour ranging from complete spreading of the lens, to spreading followed by retraction, to sustained pulsating oscillations.

[1]EPSRC Grant number EP/E046029/1

3:28PM D20.00007 Convective rolls and hydrothermal waves in evaporating sessile drops[1] , GEORGE KARAPETSAS, Imperial College London, PRASHANT VALLURI, KHELLIL SEFIANE, University of Edinburgh, OMAR MATAR, Imperial College London — We examine the dynamics of a sessile droplet sitting on a uniformly heated solid, undergoing evaporation. We use a finite element formulation to solve the axisymmetric time-dependent problem and perform a linear stability analysis taking into account 3D perturbations around this base-state. Our numerical results show that the temperature gradients, which are a natural consequence of the evaporation process, give rise to instabilities including the presence of hydrothermal waves in close agreement with previous experimental observation. We discuss the mechanisms which give rise to these instabilities.

[1]EPSRC Grant number EP/E046029/1

3:41PM D20.00008 Electrostatically-induced complex dynamics in viscous liquid film flows down the exterior of a cylinder[1] , ALEXANDER WRAY, DEMETRIOS PAPAGEORGIOU, OMAR MATAR, Imperial College London — Annular flows on both the inner and outer walls of cylinders have a significant number of practical applications, from printing to fluid stabilisation to the augmentation of heat and mass transfer. It is important to understand the spatiotemporal dynamics of the interface in order to facilitate the possible control of the flow and efficiency of underlying processes. We investigate the evolution and stability of a viscous fluid layer wetting the surface of a cylinder and surrounded by a conductive gas. The inner cylinder is an electrode kept at constant voltage. A second concentric electrode, whose potential is allowed to vary as a function of both space and time, encloses the system. This induces electrostatic forces at the interface in competition to surface tension and viscous stresses. Asymptotic methods are used to derive several systems governing the interfacial position and charge accumulation. Two canonical sets of equations are derived, one valid for long-wave systems of moderate conductivity (a coupled Craster-Matar type system) and one valid for thin films of high conductivity (a Benney-type system). The resulting systems and their stabilities are investigated both analytically and numerically to compute travelling wave and transient simulations.

[1]EPSRC Grant number EP/E046029/1

3:54PM D20.00009 An aerodynamic mechanism for the splashing-up of the ejecta sheet in high-velocity impacts , GILOU AGBAGLAH, CHRISTOPHE JOSSERAND, Institut d'Alembert CNRS-UPMC, ANDRÉA PROSPERETTI[1], Johns Hopkins University, STÉPHANE ZALESKI, Institut d'Alembert CNRS-UPMC, INSTITUT D'ALEMBERT CNRS-UPMC TEAM, DEPARTMENT OF MECHANICAL ENGINEERING, JOHNS HOPKINS UNIVERSITY COLLABORATION[2] — In this work, the deformation of a 2D liquid jet by aerodynamic stresses has been investigated by numerical simulations with the flow solver Gerris and a dynamic model was proposed to predict the shape of the jet. This is similar to the initial stages of the formation of the corolla upon impact of a drop on a solid. A self-similar behavior is observed in the simulations and the jet profile is in good agreement with the model prediction in exponential of the square root of time.

[1]University of Twente
[2]University of Twente

4:07PM D20.00010 Electric field induced instabilities at fluid-fluid interfaces in low conductivity, low frequency and small feature size limit , PRIYA GAMBHIRE, ROCHISH THAOKAR, Indian Institute of Technology, Bombay — Electrohydrodynamic instabilities at fluid-fluid interfaces due to an externally applied electric field have become a popular field of research owing to their direct application in the field of lithography. With the advent of miniaturization, it has become critical to find new ways to reduce feature sizes to even smaller dimensions and in a cost-effective way. Researchers have been working on freezing the patterns formed when thin polymer films are subjected to electric fields, which give as low as 100 nm sized structures. Different fluids with lower surface tensions and higher conductivities are being pursued to reduce the dimensions of the pattern further. Simple linear models exist which can predict these dimensions while the final structure can be simulated. The present work addresses three cases where the linear models fail. The first case is when the thin film approximation which is a common assumption in the linear theory, becomes invalid. The second, when the charge relaxation time of the fluid is of the same order as the growth rate of the instability and the third, when the time period of the applied Alternating Current (AC) field is larger than the growth rate. In all these three cases, the regime in which the linear model fails is identified and an alternate linear model or non-linear analysis has been used to predict the observed experimental behavior.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D21 Vortex Dynamics II 324-325

2:10PM D21.00001 A mathematical model of laminar wakes with four vortices per period ,

SAIKAT BASU, MARK STREMLER, Virginia Tech — Laminar wakes behind vortex-shedding bluff bodies can exhibit a variety of patterns when the bodies oscillate or are in close proximity to one another. In many cases these patterns consist of regular groupings of four vortices. We present a two-dimensional potential flow model that assumes a spatially periodic arrangement of point vortices. This Hamiltonian system is made integrable by imposing an additional spatial symmetry that is motivated by the experimental wakes. The current model generalizes our previous results by allowing for unequal vortex strengths. The model shows a variety of dynamic modes that we classify using a bifurcation analysis of the phase space structure. Some initial conditions lead to relative equilibria in which the vortex configuration moves without change in size or shape. Scaled comparisons of the model with experiments show good results and allow for estimation of the experimental vortex strengths.

2:23PM D21.00002 Numerical Investigations of Reconnection of Quantized Vortices ,

CECILIA RORAI, University of Maryland-Università di Trieste, MICHAEL E. FISHER, DANIEL P. LATHROP, University of Maryland, KATEPALLI R. SREENIVASAN, New York University, ROBERT M. KERR, University of Warwick — Reconnection of quantized vortices in superfluid helium was conjectured by Feynman in 1955, and first observed experimentally by Bewley et al. (PNAS 105, 13708, 2007). The nature of this phenomenon is quantum mechanical, involving atomically thin vortex cores. At the same time, this phenomenon influences the large scale dynamics, since a tangle of vortices can change topology through reconnection and evolve in time. Numerically, the Gross-Pitaevskii (GP) equation allows detailed predictions of vortex reconnection as first shown by Koplik and Levine (1993). We have undertaken further calculations to characterize the dynamics of isolated reconnection events. Initial conditions have been analyzed carefully, different geometries have been considered and a new approach has been proposed. This approach consists in using the diffusion equation associated to the GP equation to set minimum energy initial vortex profiles. The underlying questions we wish to answer are the universality of vortex reconnection and its effect on energy dissipation to the phonon field.

2:36PM D21.00003 Cascade of vortex tube collisions at $Re_\Gamma = 10\,000$,

WIM VAN REES, ETH Zurich, FAZLE HUSSAIN, University of Houston, PETROS KOUMOUTSAKOS, ETH Zurich — We present simulations of the collision of two anti-parallel vortex tubes, with and without axial flow in a periodic box at $Re_\Gamma = 10\,000$ using a remeshed vortex method. In the non-axial flow case, after the first, well-known vortex reconnection of the tubes, a quiescent period is followed by a second vortex collision of the remaining structures. The characteristics of this second collision are an increase of energy in the small scales of the flow; remnant vorticity left behind in thread-like structures; a persistent $-7/3$ slope in the three-dimensional energy spectrum; and a significant increase in enstrophy and helicity in the flow. Characteristics of the secondary collision are also observed during the first reconnection of the vortex tubes with axial flow. The simulations indicate that vortical flows containing initially large-scale vortical structures can transfer energy from large scales to smaller scales through a cascade of vortex collisions.

2:49PM D21.00004 Merger of multiple vortices ,

ROHITH V SWAMINATHAN, RAMA GOVINDARAJAN, Engineering Mechanics Unit, JNCASR, Bangalore, India — We study the merger of three or more identical co-rotating vortices initially arranged on the vertices of a regular polygon, and compare it to the merger of two like-signed vortices. The latter is a well studied problem, with the merger process there consisting of four stages. In the multiple (three or more) vortex case, we find a new stage in the merger process, where an annular vortical structure is formed and is long-lived. We find that merger on the whole is slowed down significantly as the number of vortices goes up, and the formation of the annular structure is primarily responsible for the delaying of the merger. Vortices initially elongate radially, and then reorient their long axis closer to the azimuthal direction, and then diffuse out to form an annulus. The inviscid case is similar at short times, but at longer times, rather pronounced filaments are visible, which are practically absent in the viscous case. The formation of an annular structure is further impeded since the azimuthal alignment is reduced. The annular stage will be contrasted with the "second diffusive stage" in two-vortex merger.

3:02PM D21.00005 Formation number of particle-laden starting jets ,

NIRANJAN GHAISAS, DINESH SHETTY, STEVEN FRANKEL, School of Mechanical Engineering, Purdue University — The dynamics of a starting jet is studied under conditions where the injected fluid is laden with small spherical particles. The pinch-off process and its associated time scale, the formation number, are studied via a series of two-way coupled particle-laden, large eddy simulations with Lagrangian tracking of the order of 10^5 particles. The particles are small enough for the point-particle approximation to be valid, and inter-particle forces are neglected since the particle to injected fluid volume fraction is smaller than 10^{-3}. Forces acting on the particle include drag force and gravitational force. The numerical code is validated by reproducing formation numbers for pure as well as positively and negatively buoyant starting jets. A systematic study is carried out to investigate the effect of particle size and density on the formation number. Results at Re=5000 indicate that the presence of particles 10 times heavier than the fluid results in significantly enhanced total and head vortex circulations, leading to formation numbers distinct from the pure starting jet case. A particle-laden injection is thus shown to behave similar to an injection of a fluid heavier than the ambient.

3:15PM D21.00006 LBM Simulations for the Head-on Collisions of Vortex Pairs and Vortices-Wall Interactions in Two Dimensions[1] ,

YUXIAN XIA, Institute of Applied Mathematics and Mechanics, Shanghai University, Shanghai 200072, YUEHONG QIAN, Institute of Applied Mathematics and Mechanics, Shanghaqi University, Shanghai, 200072 — Vortex dynamics has been studied in two dimensions with lattice Botzmann method for the basic interactions of vortices. With different off-center distances the head-on collisions of two equal pairs of vortices(two dipoles) different regimes have been found: exchanging, merging and diffracting. Meanwhile the interactions of a vortex dipole with solid wall have been also investigated and comparison with existing experiments has been shown certain agreement. More complicated cases for three dimensions are left for future works.

[1]This research is support in part by the grant of Education Ministry of China IRT0844 and the grant of Shanghai CST 11XD1402300 and that of Leading Scientists.

3:28PM D21.00007 Investigation of the critical strain rate for co-rotating vortex pairs ,

PATRICK FOLZ, KEIKO NOMURA, University of California, San Diego — Two-dimensional interactions of a co-rotating vortex pair in a viscous fluid are investigated. With symmetric vortex pairs, i.e. those with two vortices of equal size and strength (Brandt & Nomura, *J. Fluid Mech.*, vol. 592, 2007), the mutually-induced strain deforms and tilts the vortices, which leads to a core detrainment process. The weakened vortices are mutually entrained and rapidly move towards each other as they intertwine and destruct, thereby developing into a single compound vortex. With asymmetric pairs, the interactions are more complex and may result in different outcomes. These can be classified according to the relative timing of core detrainment and core destruction of the vortices. A critical strain rate parameter which characterizes the establishment of core detrainment has been identified and determined. In rudimentary studies, this critical value was found to be consistent for a range of vortex strength ratios (Brandt & Nomura, *J. Fluid Mech.*, vol. 646, 2010). In the present study, further analysis and numerical simulations are performed to ascertain the degree of universality of the critical strain rate value. A larger range of flow parameters is considered. Results clarify the relationship between the parameter and the behavior of these vortex flows.

3:41PM D21.00008 Stirring, Stretching and Transport Generated by a Vortex Pair , LUCA CORTELEZZI, McGill University, FRANCESCO RIZZI, Johns Hopkins University — We consider a pair of like-signed, initially elliptical vortices with uniform vorticity distribution embedded in an incompressible, inviscid fluid occupying a two-dimensional, infinite domain. The rotational and co-rotational motion of the vortices stir the fluid and subdivide the domain into inner core, inner recirculation, outer recirculation regions and outer flow. We quantify stirring using stretching of the interface and the mix-norm. Our numerical simulations show that stretching is dominated by the chaotic advection induced within the inner core and inner recirculation regions, whereas the mixnorm is dominated by the laminar transport induced within the outer recirculation regions. Stirring is sensitive to the geometry of the initial concentration field. We consider, as an initial scalar field, two concentrations delimited by a straight-line interface of adjustable orientation and show that the interface passing through the centroids of the vortices is the one most efficiently stretched, while the initial concentration field with an orthogonal interface is the most efficiently stirred. Finally, we investigate the effects of the angular impulse on the stirring performance of the vortex pair. Stretching is very sensitive to the angular impulse, while the mix-norm is not. We show that there is a value of the angular impulse which maximizes stretching and argue that this is due to two competing mechanisms. Funding provided by NSERC, contract RGPIN217169.

3:54PM D21.00009 Aeroacoustics of viscous vortex reconnection , PEDRO PAREDES, Universidad Politecnica de Madrid, JOSEPH W. NICHOLS, KARTHIK DURAISAMY, Stanford U., FAZLE HUSSAIN, U. of Houston — Reconnection of two anti-parallel vortex tubes is studied by direct numerical simulations and large-eddy simulations of the incompressible Navier-Stokes equations over a wide range (2000-50,000) of the vortex Reynolds number (Re). A detailed investigation of the flow dynamics is performed and at high Re, multiple reconnections are observed as the newly formed "bridges" interact by self and mutual induction. To investigate acoustics produced by the recoil action of the vortex threads, Möhring's theory of vortex sound is applied to the flow field and evaluated at varying far-field locations. The acoustic solver is verified against calculations of laminar vortex ring collision. For anti-parallel vortex reconnection, the resulting far-field spectra are shown to be grid converged at low-to-mid frequencies. To assess the relevance to fully turbulent jet noise, the dependence of reconnection upon Reynolds number is investigated.

4:07PM D21.00010 Formation regimes of vortex rings in negatively buoyant starting jets , CAROLINA MARUGAN-CRUZ, JAVIER RODRIGUEZ-RODRIGUEZ, Carlos III Univ. of Madrid, CARLOS MARTINEZ-BAZAN, Univ. of Jaen — The formation of vortex rings (VR) in negatively buoyant starting jets has been studied numerically for different values of the Richardson number covering the range of weak to moderate buoyancy effects ($0 \leq \mathrm{Ri} < 0.20$). We have identified two different regimes in the vortex formation whose transition takes place at about $\mathrm{Ri} \approx 0.03$. The vorticity distribution inside the VR after pinching-off as well as the total amount of circulation it encloses (characterized by the formation number, F) show different behaviors with Ri in both regimes. Nevertheless, the physical mechanism limiting the circulation of the vortex, or equivalently the formation number, is the same in both cases. Thus, the formation number of a negatively buoyant VR, whose propagating velocity is slower than that of a neutrally buoyant one due to gravity effects, can be determined considering that it is nearly the same as that of a neutrally buoyant VR moving with the velocity corresponding to the negatively buoyant vortex. Based on this simple idea, a phenomenological model is presented to quantitatively describe the evolution of the formation number with the Richardson number, $\mathrm{F(Ri)}$, obtained numerically. We also discuss the limitations of different vortex identification to evaluate the vortex properties in buoyant flows. Supported by the Spanish Ministry of Science grant DPI2008-06369.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D23 Reacting Flows I: Turbulent Combustion 326

2:10PM D23.00001 Role of Large Scale Mixing in Soot Evolution in Turbulent Nonpremixed Combustion , MICHAEL MUELLER, Stanford University, HEINZ PITSCH, Stanford University; RWTH Aachen University — In order for soot to form and grow, long residence times are required at rich mixture fractions. Under these conditions, soot particles grow either by surface reactions with acetylene or by condensation of Polycyclic Aromatic Hydrocarbons (PAH) onto the surface of the particles. In past studies on nonpremixed combustion, condensation of PAH was found to be the dominant growth mechanism, while surface reactions with acetylene played only a minor role. In this work, a recently developed LES model for soot evolution in turbulent nonpremixed combustion is used to examine two configurations in which long residence times under rich conditions are attained in distinctly different ways. In the first configuration, a typical piloted jet flame, soot is formed far downstream from the burner nozzle and grows primarily by PAH condensation. In the second configuration, a bluff body stabilized flame, the dominant soot growth mechanism is found to be surface growth by acetylene. Here, the recirculation zone formed behind the bluff body provides the long residence times needed for soot formation. The combination of very small dissipation rates and a narrow band of mixture fractions in the recirculation zone promotes surface growth by acetylene over PAH condensation.

2:23PM D23.00002 Simultaneous soot concentration and strain-rate measurements in turbulent jet flames , VENKAT NARAYANASWAMY, Institute for Combustion Technology, RWTH Aachen, NOEL CLEMENS, The University of Texas at Austin, Austin, TX — Studies on soot formation are very important both from an environmental standpoint and from a technical perspective. The correlation between the instantaneous soot concentration fields and the corresponding instantaneous strain fields, which would provide important details on the fluid-dynamic aspects of soot formation, relevant for practical combustors, is not understood yet. To address this issue, experiments were performed in our laboratory to study the organization of 2-D soot concentration fields, obtained using LII, and its correlation with corresponding instantaneous 2D strain-rate fields, obtained simultaneously using PIV. The experiments were performed in a turbulent co-flowing jet facility, with an ethylene/N_2 mixture as the fuel. Different jet-exit Reynolds numbers were obtained by changing the jet-exit velocity, and the corresponding evolution of the soot and strain-field structures is investigated. Our preliminary results show that the instantaneous soot field topology is highly correlated with the instantaneous strain-rate topology. The regions of intense soot concentrations are mainly along the regions of intense strain-rate; furthermore, the soot concentration fields become increasingly convoluted and sparse with increasing Reynolds number.

2:36PM D23.00003 Direct numerical simulation of temporally evolving turbulent luminous jet flames with detailed fuel and soot chemistry , VIVIEN LECOUSTRE, University of Maryland, PAUL ARIAS, University of Michigan, SOMESH ROY, Pennsylvania State University, WEI WANG, University of Tennessee, ZHAOYU LUO, University of Connecticut, DAN HAWORTH, Pennsylvania State University, HONG IM, University of Michigan, TIANFENG LU, University of Connecticut, KWAN-LIU MA, University of California-Davis, RAMANAN SANKARAN, Oak Ridge National Laboratory, ARNAUD TROUVE, University of Maryland — Direct numerical simulations of 2D temporally-evolving luminous turbulent ethylene-air jet diffusion flames are performed using a high-order compressible Navier-Stokes solver. The simulations use a reduced mechanism derived from a detailed ethylene-air chemical kinetic mechanism that includes the reaction pathways for the formation of polycyclic aromatic hydrocarbons. The gas-phase chemistry is coupled with a detailed soot particle model based on the method of moments with interpolative closure that accounts for soot nucleation, coagulation, surface growth through HACA mechanism, and oxidation. Radiative heat transfer of CO_2, H_2O, and soot is treated by solving the radiative transfer equation using the discrete transfer method. This work presents preliminary results of radiation effects on soot dynamics at the tip of a jet diffusion flame with a particular focus on soot formation/oxidation.

2:49PM D23.00004 Turbulent propagation of premixed flames: the effects of thermal expansion and integral scale , FRANCESCO CRETA, University of Rome La Sapienza, MOSHE MATALON, University of Illinois at Urbana-Champaign — We study the propagation of a premixed flame in a turbulent environment within the wrinkled flamelet regime. We adopt a hybrid Navier-Stokes/front capturing technique whereby the flame is assimilated to a surface of vanishing thickness separating burnt and unburned regions and subjected to an incident two-dimensional turbulent flow. Among the many observable phenomena, two separate effects on the turbulent propagation speed are investigated, namely that of the thermal expansion (unburned to burnt gas density ratio) and of turbulence integral scale. Turbulent speed is observed to increase and later plateau as thermal expansion increases, this being due to an increase in flame brush thickness with no perceivable curvature variation. The effect of the integral scale of the incoming turbulent field reveals the existence of a particular intermediate value of such scale at which the flame experiences a maximum propagation speed. The same qualitative tendencies are confirmed by a weakly nonlinear Michelson Sivashinsky (MS) model forced with spatially correlated noise that mimics the incoming turbulent flow. Both effects can be combined, together with the effect of turbulence intensity, in a general scaling law for the turbulent propagation speed valid for hydrodynamically stable flames.

3:02PM D23.00005 Turbulent Flame Speed and Self Similarity of Expanding Premixed Flames , SWETAPROVO CHAUDHURI, FUJIA WU, DELIN ZHU, CHUNG LAW, Princeton University — In this study we present experimental turbulent flame speed data measured in constant-pressure expanding turbulent flames, propagating in nearly homogenous isotropic turbulence, in a dual-chamber, fan-stirred vessel. The cold flow is characterized by high speed particle image velocimetry while the flame propagation rate is obtained by tracking high speed Schlieren images of unity Lewis number methane-air flames over wide ranges of pressure and turbulence intensity. It is found that the normalized turbulent flame speed as a function of the average radius scales as a turbulent Reynolds number to the one-half power, where the average radius is the length scale and thermal diffusivity is the transport property, thus showing self-similar propagation. Utilizing this dependence it is found that the turbulent flame speeds from expanding flames and those from Bunsen geometries can be scaled by a single parameter: the turbulent Reynolds number utilizing recent theoretical results obtained by spectral closure of the G equation, after correcting for gas expansion effects.

3:15PM D23.00006 An extinction/reignition dynamic method for turbulent combustion , ROBERT KNAUS, CARLOS PANTANO, University of Illinois at Urbana-Champaign — Quasi-randomly distributed locations of high strain in turbulent combustion can cause a nonpremixed or partially premixed flame to develop local regions of extinction called "flame holes". The presence and extent of these holes can increase certain pollutants and reduce the amount of fuel burned. Accurately modeling the dynamics of these interacting regions can improve the accuracy of combustion simulations by effectively incorporating finite-rate chemistry effects. In the proposed method, the flame hole state is characterized by a progress variable that nominally exists on the stoichiometric surface. The evolution of this field is governed by a partial-differential equation embedded in the time-dependent two-manifold of the flame surface. This equation includes advection, propagation, and flame hole formation (flame hole healing or collapse is accounted by propagation naturally). We present a computational algorithm that solves this equation by embedding it in the usual three-dimensional space. A piece-wise parabolic WENO scheme combined with a compression algorithm are used to evolve the flame hole progress variable. A key aspect of the method is the extension of the surface data to the three-dimensional space in an efficient manner. We present results of this method applied to canonical turbulent combusting flows where the flame holes interact and describe their statistics.

3:28PM D23.00007 Intermittency in Premixed Turbulent Reacting Flows[1] , PETER HAMLINGTON, ALEXEI POLUDNENKO, ELAINE ORAN, Naval Research Laboratory — Characterizing the intermittency of velocity gradient and scalar gradient fields in turbulent reacting flows is important for developing a better understanding of the interactions between turbulence and flames. Here we examine intermittency in premixed reacting flows using numerical simulations of stoichiometric hydrogen-air combustion at a range of turbulence intensities. Simulations of homogeneous isotropic turbulence with a nonreacting passive scalar are also carried out in order to allow comparisons with the reacting flow results. We examine intermittency by calculating probability density functions and moments of the local enstrophy, energy dissipation rate, and scalar dissipation rate. Conditional analyses based on local, instantaneous values of the reactant mass fraction are used to study variations in the statistics through the flame. We observe variations in the intermittency depending on the intensity of the turbulence, the location in the flame, and the quantity under consideration. We discuss the implications of these results for the flame structure, and also provide an explanation for the observed results by considering the two-way interactions between turbulence and premixed flames.

[1]This research was performed while PEH held a National Research Council Research Associateship Award at the Naval Research Laboratory.

3:41PM D23.00008 Using LCS to study coherent structures in reacting flows[1] , MELISSA GREEN, PETER HAMLINGTON, ALEXEI POLUDNENKO, ELAINE ORAN, Naval Research Laboratory — Previous research has shown that chemical reactions in a compressible fluid flow interact strongly with the surrounding turbulence both in quantitative measures and qualitative character. In the case of a flame propagating through homogeneous isotropic turbulence, the rapid flow expansion generated in the reaction zone causes a significant attenuation in the vorticity. This suppression of the vorticity magnitude complicates the tracking of individual coherent structures using Eulerian methods, therefore we use Lagrangian coherent structures to study the nature of the vortex dynamics, focusing on structure creation, destruction, and reorientation.

[1]This research was performed while the author held a National Research Council Research Associateship Award at the Naval Research Laboratory

3:54PM D23.00009 *A Priori* Analysis of Subgrid Mass Flux Vectors from Massively Parallel Direct Numerical Simulations of High Pressure H2/O2 Reacting Shear Layers[1] , JUSTIN FOSTER, RICHARD MILLER, Clemson University — Direct Numerical Simulations (DNS) are conducted for temporally developing reacting H2/O2 shear layers at an ambient pressure of 100atm. The compressible form of the governing equations are coupled with the Peng Robinson real gas equation of state and are solved using eighth order central finite differences and fourth order Runge Kutta time integration with resolutions up to $\sim$3/4 billion grid points. The formulation includes a detailed pressure dependent kinetics mechanism having 8 species and 19 steps, detailed property models, and generalized forms of the multicomponent heat and mass diffusion vectors derived from nonequilibrium thermodynamics and fluctuation theory. The DNS is performed over a range of Reynolds numbers up to 4500 based on the free stream velocity difference and initial vorticity thickness. The results are then analyzed in an *a priori* manner to illustrate the role of the subgrid mass flux vector within the filtered form of the governing equations relevant to Large Eddy Simulations. The subgrid mass flux vector is found to be a significant term; particularly within localized regions of the flame.

[1]Research supported by NSF Grant CBET-0965624 and Clemson University's Palmetto Cluster.

4:07PM D23.00010 Characteristics of Turbulent Premixed Flames under the Pressure Rising Process in a Closed Vessel , NAOYA FUKUSHIMA, BASMIL YENERDAG, MASAYASU SHIMURA, MAMORU TANAHASHI, TOSHIO MIYAUCHI, Tokyo Institute of Technology — In a closed vessel such as SI engines, the internal pressure increases due to dilatation during the combustion after the ignition. To clarify quantitative characteristics of turbulent premixed flames under the pressure rising process, direct numerical simulation (DNS) of turbulent premixed flames in a closed vessel at relatively high Reynolds number has been conducted. Detailed kinetic mechanism for hydrogen-air mixtures is used. Because of the local pressure rise, turbulence is enhanced at the unburnt side and flame surface is distorted, which results in increase of the flame surface. Heat release rate of each flame element is augmented since the pressure rise makes flame thickness thin. Under this pressure rising process, the flame thickness, the flame front curvature and the local heat release rate can be scaled by laminar flame thickness and the maximum heat release rate obtained from one dimensional DNS of laminar flame propagation by using averaged temperature in the unburnt region of the vessel as the inlet temperature. The tangential strain rate on the flame front can be scaled by Taylor micro scale averaged in the unburnt side. The local heat release rate is positively correlated with the curvature and the tangential strain. The time evolution of the flame surface area is also investigated quantitatively.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D24 Particle-laden Flows I 327

2:10PM D24.00001 Physical perspectives on the investigation of two-way interaction in particle-laden isotropic turbulence , ABOU-ELMAGD ABDEL-SAMIE, CHANGHOON LEE, Yonsei University — The two-way coupling interaction in an isotropic turbulence has been investigated in both decaying and stationary turbulence to examine the effect of artificial forcing for the maintenance of stationary turbulence, on turbulence modulation mechanism. Direct numerical simulations (DNS) for stationary and decaying isotropic turbulence have been carried out using 128^3 grids with the Taylor micro-scale, $R_\lambda \simeq 70$, in the presence of 10^6 solid sphere particles whose diameter is smaller than the Kolmogorov length scale. The particles which were released with different Stokes number ($0 < St < 5$), are implemented as a point force approximation in Navier-Stokes equation. Turbulence kinetic energy, acceleration, enstrophy and their spectra have been examined to display the distinctions between decaying and stationary turbulence. A mathematical analysis is provided to support our physical perspectives, where we argued that the stationary turbulence is not appropriate for the study of turbulence modulation by particles with $St < 1$. Furthermore, it is shown that the injection perturbation of the particle, due to coupling, has a significant effect on the turbulence modulation in decaying turbulence. An investigation of correlation between turbulence field zones and the two-way interaction energy has been conducted for a better understanding of the modulation mechanism.

2:23PM D24.00002 Acceleration modification of near-wall turbulence by heavy particles , JUNGHOON LEE, CHANGHOON LEE, Yonsei University — By conducting direct numerical simulation combined with Lagrangian particle tracking, we examine modifications of the near-wall turbulence by heavy particles. For simplicity, only spherical and rigid particles are considered in our study and particles are idealized as point-sources in our spectral simulation of turbulent channel flow laden with heavy particles. To assess the effect of particles on the flow field, we use the Particle-Source In Cell (PSIC) model proposed by Crowe et al (1977). Because particles suspended in wall-bounded turbulence can modify the near-wall turbulent structures depending on Stokes number, and these structures are associated with the intermittent, strong acceleration of fluid, particles are expected to modify the turbulence acceleration statistics in the near-wall region. Therefore, in the present study, we especially focus on the modification of the acceleration statistics of turbulence by particles in turbulent channel flow for various Stokes numbers. Detailed statistical changes of acceleration of turbulence by particles and plausible physical explanations will be presented in the meeting.

2:36PM D24.00003 The effect of particle size on the dynamics of a solid particle in a turbulent carrier flow , HUI GAO, ORLANDO AYALA, LIAN-PING WANG, University of Delaware — When the particle size is much smaller than the Kolmogorov scale of the carrier flow, the motion of the particle can be described by a point-particle model. Currently, it is not clear how to treat the motion of a solid particle when its size is comparable or larger than the Kolmogorov scale. Here we address this by using a numerical solution from a particle-resolved simulation code for a freely-moving finite-size particle suspended in a turbulent background flow. The code is based on the multiple-relaxation-time lattice Boltzmann equation. The no-slip boundary condition on the moving particle boundary is handled by a second-order interpolated bounce-back scheme. The populations at a newly converted fluid node are constructed by the equilibrium distribution with non-equilibrium correction. The main purpose here is to extend some recent advances made regarding the finite size effects on a buoyant particle to finite-size heavy or light particles where both inertial effect and finite-size effect are important. We will present results for different particle sizes relative to the Kolmogorov (or Taylor) microscale and different particle to fluid density ratios. The force acting on the particle and particle velocity and acceleration statistics will be discussed.

2:49PM D24.00004 Highly-scalable simulation of turbulent and particle-laden flows using the lattice Boltzmann approach , CHARLES ANDERSEN, ORLANDO AYALA, HUI GAO, LIAN-PING WANG, University of Delaware — Particle-laden turbulent flows are found in many applications such as sediment transport, pollutant dispersion, interaction of cloud droplets, and chemical processing. Recently, we have developed a particle-resolved direct simulation of particle-laden turbulent flow using the mesoscopic lattice Boltzmann approach. Here we explore the parallel scalability of this approach by comparing the parallel efficiencies of the code using one, two, and three dimensional domain decompositions. First, we compare the scalability data of the turbulent flow simulation without particles and show that the communication overhead in the approach is negligible due to the local nature of data communication. Then, we discuss parallel implementation issues related to the interaction of solid particles with the carrier turbulent flow. The new results based on multiple dimensional decompositions will be compared to the previous results using one dimensional domain decomposition. Results from high-resolution particle-resolved multiphase flow simulations will be used to address flow modulation by finite-size solid particles when the size of the particles is comparable to or larger than the flow Kolmogorov scale.

3:02PM D24.00005 Development of modified body-force type immersed boundary method , TOMOYA WAKAMATSU, TAKUYA TSUJI, TOSHITSUGU TANAKA, Osaka University — Particulate flows occur frequently in nature and engineering applications. Examples include fluidized bed, transportation of aerosols, blood flow and so on. The behavior of flows is highly complex and it is still difficult to obtain reliable information experimentally and a computational prediction is required especially for the microscopic flows in the particle-level. In recent years, a number of CFD technique for fluid-solid interaction problems have been proposed. Some examples are immersed boundary method, lattice Boltzman and fictitions domain method. "Body-force type immersed boundary method" originally developed by Kajishima et al. (2001) is a simple scheme and have been applied to several problems. The method is based on a fixed Cartesian grid and solid body is represented by its volume fraction at each calculation cell. The method is quite efficient while it has intrinsic problems such as Δt-dependency of calculation results and the violation of divergence-free condition. In the present study, these problems are revealed and a modified method is proposed. Computations based on the present method are performed for a number of fluid-particle interaction problems and results are compared with the original one.

3:15PM D24.00006 Particle-wall impacts in a confined extensional flow , DANIELE VIGOLO, Princeton University, IAN GRIFFITHS, ANTHONY LOCK, University of Oxford, HOWARD STONE, Princeton University — Buoyant particles entrained in liquid flowing in confined geometries such as pipes and channels arise in a broad spectrum of areas including engineering and the natural and biological sciences. Understanding the particle behavior upon changes in flow direction is crucial in problems where particle inertia is important, such as the erosion process in pipe bends. We present results on the impact of particles in a T-shaped channel in the laminar-turbulent transitional regime. The impacting event for a given system regime is described in terms of the particle Stokes number and the Reynolds number, where for the model local extensional flow the latter also characterizes the ratio of particle size to thickness of the viscous boundary layer which forms in the region below the impingement. Experimental results for the impact are compared with the trajectories predicted by theoretical particle-tracing models for a range of configurations to determine the role of the viscous boundary layer in slowing down the particles and reducing the rate of collision with the substrate. The implications of our results on the erosion process in a piping system are discussed.

3:28PM D24.00007 Rotational Diffusion of Particles in Turbulence , EVAN VARIANO, COLIN MEYER, MARGARET BYRON, University of California, Berkeley — We experimentally compare the rotation of spherical and ellipsoidal particles in homogeneous, isotropic turbulence. We find that the particle orientation is well described by a Gaussian diffusion process. This theoretical model would predict that the Lagrangian autocorrelation function for angular velocity is a negative exponential. We measure this Lagrangian autocorrelation function using stereoscopic particle image velocimetry (SPIV) applied to particles whose size is within the inertial subrange of the ambient turbulence. The SPIV resolves 3 velocity components in a nearly 2-dimensional planar volume, which we use as inputs for a nonlinear optimization to quantify the solid body rotation of the particles. This provides us the angular velocity timeseries for individual particles. Through ensemble statistics, we determine the Lagrangian autocorrelation function of angular velocity, from which we can quantify the turbulent rotational diffusivity and its behavior between the extremes of short-term non-Fickian transport and long-term Fickian diffusion.

3:41PM D24.00008 Dynamics of Arrays of Falling Cylinders and Modeling of Collisions Using the Lubrication Theory , AÇMAE EL YACOUBI, Cornell University, SHENG XU, Southern Methodist University, Z. JANE WANG, Cornell University — Motivated by our interest in understanding collective behavior and self-organization resulting from hydrodynamic interactions, we investigate the dynamics of horizontal arrays of settling cylinders at intermediate Re. Here, we focus on pairwise interactions by quantifying the interaction force for a pair of cylinders and its dependence on the initial spacing d_0. We fiund that the pair initially experiences a repulsive force which varies as $1/d_0^\alpha,\ \alpha > 0$. We then study the dynamics of a falling cylinder in presence of its left and right neighbors and compare them to those of an isolated cylinder and with results in Stokes flow. Our findings show that, unlike in Stokes flow, the middle cylinder experiences a higher drag force due to the presence of co-moving cylinders, resulting in a slower settling velocity. Additionally, the smaller d_0, the sooner wake asymmetries for the middle cylinder arise and the sooner it settles to steady-state. For small d_0 or for large collections of cylinders ($n > 5$), collisions occur. Instead of using an $ad\ hoc$ collision model, we derive the interstitial pressure field using the lubrication theory. The lubrication pressure and shear force are then used to compute the repulsive force during the approach phase between two particles. Results using this theory will be compared to those using a simple dry collision model.

3:54PM D24.00009 Monodisperse and Polydisperse Particle Flow over a Backward Facing Step Preceding a Porous Medium , FRANK CHAMBERS, ALOK DANGE, Oklahoma State University — Computational Fluid Dynamic predictions were performed for the flow of monodisperse and polydisperse particles over a backward facing step with and without a porous medium downstream. The carrier fluid was air and the particles had a density of 500 kg/m3. Monodisperse particles with diameters of 10 and 40 microns and polydispersed particles from 1 to 50 microns with a Rosin-Rammler size distribution were used. The step had an expansion ratio of 2 and the step Reynolds numbers were 6550 and 10000. The k-epsilon RNG model with standard wall functions was used with FLUENT's discrete phase model for the particles. Velocity and particle residence time tracks were examined. The placement of the medium at 4.25h from the step was found to control the velocity profiles and the length of the recirculation zone while placement at 6.75h had negligible effects. The particle tracks show that more particles with lower Stokes number enter the recirculation zone while the particles with higher Stokes number tend to bypass the recirculation zone and move directly to the filter. The results for the monodispersed and the polydispersed particles appear virtually the same at low particle concentrations, but the polydispersed results provide a very good view of the phenomena.

4:07PM D24.00010 Experimental investigation of free falling thin disks. Part I: The flow structures and the Reynolds number effects of zigzag motion , CUNBIAO LEE, College of Engineering, Peking University, HONGJIE ZHONG — We present experimental investigations on the behaviors of free falling thin circular disk in still water. Flow patterns of the zigzag disk motion are studied with dye visualization and particle image velocimetry. Time-resolved disk motions with six degrees of freedom are obtained with a stereoscopic vision method. The flow separation and shedding vortices are found changing with free falling Reynolds number Re. At high Reynolds number, we found a new dipole vortices shedding from the disk besides the Karman-type vortices at low Reynolds numbers. The vortical structures are mainly composed of leading edge vortices, counter-rotating vortex pair and secondary trailing edge vortices. It was found that the dimensionless amplitude of horizontal oscillation was dependent on the Reynolds number Re. There existed a critical Reynolds number about 2000, oscillatory amplitude was proportional to Re when Re is below the critical value, but invariant with Re when Re is above this value. The onset of dipole vortices was directly related to the increasing Reynolds number and horizontal oscillations.

Sunday, November 20, 2011 2:10PM - 4:07PM –
Session D25 Supersonic and Hypersonic Flows 328

2:10PM D25.00001 Respirable Particle Transport from Surfaces by Shock Waves[1] , C.R. TRUMAN, P. VOROBIEFF, J. CONROY, P. WAYNE, R. WHITE, M. ANDERSON, U. New Mexico, S. KUMAR, U. Texas, Brownsville — Resuspension of particles from planar surfaces was studied in a shock tube. Respirable particles (aerodyn. diam.$\leq 5\ \mu$m) and slightly larger non-respirable particles were tested on smooth and rough surfaces at Mach 1.2 to 2.0. Particles of specified size were deposited on substrates of prescribed roughness. Surface roughness and particle-surface adhesion forces were quantified by atomic force microscopy. Alkylthiol self assembled monolayers (SAMs) were applied to precisely control surface roughness and surface chemistry. The advection of particles initially at rest on the surface by the rapidly accelerated flow were measured by Mie scattering. An ultra-high-speed digital camera with pulsed laser sheet illumination enables time-resolved particle transport diagnostics. Although particles are initially swept off a smooth surface with greater ease, cloud propagation speed is higher for a rough surface. At late times the cloud height is greater for a rough surface so that particles end up in a faster region of the boundary layer. Because our respirable and non-respirable particle size distributions overlap, further study is required. Shear-driven Kelvin-Helmholtz vortices clearly visible in some images likely play a prominent role in particle transport.

[1]Supported by DTRA Threat Agent Science, Agent Characterization.

2:23PM D25.00002 DNS of a Hypersonic Shock Wave/Turbulent Boundary Layer Interaction[1] , STEPHAN PRIEBE, JUSTINE LI, PINO MARTIN, University of Maryland, College Park — The direct numerical simulation of a hypersonic shock wave/turbulent boundary layer interaction (STBLI) generated by a 33-degree compression ramp is presented. The fully-turbulent inflow boundary layer is at Mach 7.2, and the Reynolds number based on momentum thickness is $Re_\theta = 3500$. The evolution of the mean and fluctuating field through the interaction region is investigated. In the supersonic regime, STBLI flows are known to display low-frequency unsteadiness, typically at frequencies 1-2 orders of magnitude lower than the characteristic frequency of the incoming undisturbed boundary layer. Preliminary observations are made about the properties of the low-frequency unsteadiness in the present hypersonic interaction.

[1]This research is supported by AFOSR Grant Number AF 9550-09-1-0464.

2:36PM D25.00003 High-speed boundary layer transition induced by a discrete roughness element[1] , PRAHLADH IYER, KRISHNAN MAHESH, University of Minnesota — The effect of a hemispherical bump on a Mach 3.37 laminar boundary layer is studied using DNS for three conditions with $k/\delta = 2.54$, 0.25 and 0.125, where k is the roughness height. The simulation parameters are based on the experiment by Danehy *et. al.* (AIAA-2009-394). The flow downstream of the roughness is transitional for all the three conditions accompanied by a rise in skin friction and heat transfer. Upon interaction with the roughness element, the boundary layer separates to form a series of spanwise vortices upstream and a shear layer. These vortices wrap around the roughness to yield a system of streamwise vortices downstream. Perturbation of the shear layer due to the vortices results in the formation of hairpin-shaped vortices further downstream of the roughness. While hairpin vortices were observed in both the center plane and off-symmetry planes on either side for the smallest δ case, they were observed only in the center plane for the other cases.

[1]This work was supported by NASA under the hypersonics NRA program under grant NNX08AB33A.

2:49PM D25.00004 Velocity and Scalar Measurements of Strut-Based Hypermixing Geometries in a Mach 3 Flow[1] , ROSS BURNS, NOEL CLEMENS, The University of Texas at Austin — Strut-based fuel injection with hypermixing exhibits great potential as a fuel-injection strategy for future scramjet engine design. Hypermixing entails the introduction of strong streamwise vorticity by means of geometrically-induced pressure gradients at the trailing edge of the strut; however, these complex flowfields are not well understood. An experimental investigation is being conducted on the flowfield characteristics of several strut-based hypermixers in a Mach 3 freestream. The hypermixing flowfields are generated from an injection pylon with interchangeable trailing-edge geometries including compressive and expansive wedges. Particle image velocimetry (PIV) in conjunction with two scalar visualization techniques are used to obtain velocity and scalar field data. The scalar imaging techniques include two-photon absorption planar laser-induced fluorescence (PLIF) of krypton gas, which simulates fuel injection into the wake, and planar laser scattering (PLS) from condensed carbon dioxide fog, which marks the outer flow structures. The velocity and scalar data reveal details of the underlying flow physics as well as the turbulent mixing characteristics.

[1]This work was supported by NASA under cooperative agreement NNX08AB41A.

3:02PM D25.00005 Thermal Loads on a Domed Protuberance Under a Mach 5.7 Boundary Layer[1] , CHRISTOPHER OSTOICH, DANIEL BODONY, PHILIPPE GEUBELLE, University of Illinois at Urbana-Champaign — A high-fidelity, high-accuracy multi-physics computational tool has been developed to make predictions of structural-thermal response in the hypersonic regime. The predicted surface heat flux distribution was compared with measured data taken from a 1986 experiment in the NASA Langley 8-foot high-temperature tunnel in which a flat plate with a domed protuberance was inserted into a Mach 6.59 flow. The solution from the fluid and thermal domains obtained from the coupled simulation, with experimental comparisons, will be presented. In addition to the typical heating associated with windward-facing surfaces, several other sources of significant differential heating were observed near the dome-plate interface and due to a trailing horseshoe vortex of small size. Data were collected over a long time record (50 seconds) and comments will be given about ignoring transient thermal effects in hypersonic boundary layer calculations. An assessment of gas thermal model assumptions will also be discussed.

[1]Supported by the U.S. Air Force Research Laboratory Air Vehicles Directorate under contract number FA8650-06-2-3620.

3:15PM D25.00006 Experimental Study of Supersonic Inlet Flow Unstart Induced by Mass Injection[1] , HYUNGROK DO, University of Notre Dame, SEONG-KYUN IM, Stanford University, MARK G. MUNGAL, Santa Clara Universtiy, MARK A. CAPPELLI, Stanford University — It is demonstrated that the boundary layer conditions of supersonic model inlet flows strongly affect the unstart that is induced by a transverse jet injection. Planar laser Rayleigh scattering from condensed CO_2 particles is utilized to visualize flow features. Studies conducted over a range of inlet configurations reveal that relatively thick turbulent boundary layers in asymmetric wall boundary layer conditions prompt the formation of oblique unstart shocks that facilitates fast inlet unstart. In contrast, thin symmetric boundary layers span pseudo-shocks which appear to be quasi-stationary under some configurations. The unstart threshold is found to be sensitive to channel height and the relative concentration of injected CO2. We find that higher jet injection pressure can be accommodated with higher CO_2 concentration because of the heavier molecular weight of CO_2, and that unstart occurs at lower jet injection rates for smaller inlet model heights.

[1]This material is based upon work supported by the Department of Energy under Award Number(s)DE-FC52-08NA28614.

3:28PM D25.00007 Unsteady Aspects of an Oblique Shock Reflection over a Heated Wall[1] , VINCENT JAUNET, PIERRE DUPONT, JEAN-PAUL DUSSAUGE, IUSTI - UMR CNRS 6595 and Aix-Marseille University — In supersonic flows, when an oblique shock wave impinges a boundary layer and makes it separate, strong aerodynamical loads at low frequency are created. This study aims at studying density effects on these structures by means of wall heating. Experiments are conducted at Mach 2.3. The temperature of the floor of the test section can be heated up to twice the recovery value. The interaction length is investigated through mean schlieren visualizations. It turns out that the interaction length increases of about 30% between the adiabatic case and the heated one, whatever the adverse pressure gradient involved. Hotwire measurements are performed in the external flow in order to characterize the unsteadiness of the reflected shock. Results show that lower frequencies are involved in the heated case, in accordance with the Strouhal number of the interaction based on the interaction length L and the external velocity U_e: $S_t = \frac{f*L}{U_e} \simeq 0.03$.

[1]The authors wish to thank the CNES for supporting this program.

3:41PM D25.00008 ABSTRACT WITHDRAWN –

3:54PM D25.00009 Bifurcation of Scramjet Unstart[1] , IK JANG, Stanford Univ., JOSEPH NICHOLS, Center for Turbulence Research, KARTHIK DURAISAMY, PARVIZ MOIN, Stanford Univ. — We investigate the bifurcation structure of catastrophic unstart in scramjets. The bifurcation of quasi-one-dimensional Rayleigh flow is first analyzed, followed by a numerical investigation of a more realistic model scramjet isolator (Wagner et al., AIAA paper, 2010). We show that the quasi-one-dimensional model recovers a similar hysteresis behavior as that observed in steady Reynolds-Averaged Navier–Stokes simulations of the model scramjet isolator close to the onset of unstart. In the hysteresis zone, steady but unstable solutions are obtained by means of pseudo-arclength continuation. Automatic differentiation permits the use of fully discrete Jacobians that result in an accurate representation of functional dependencies and linearized dynamics. Furthermore, we use an Arnoldi method to extract the least stable direct and adjoint eigenfunctions spanning the system dynamics close to unstart and obtain the system response to both harmonic and stochastic forcing. This information, along with the final bifurcation structure, allows us to evaluate the effectiveness of different metrics as indicators of the onset of unstart.

[1]Supported by the PSAAP program of DOE

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D26 Rotating Flow I 329

2:10PM D26.00001 Spin-up of a stratified fluid in a square cylinder with a sloping bottom , M.R. FOSTER, Rensselaer Polytechnic Institute, R.J. MUNRO, University of Nottingham,UK — We consider the spin-up of a linearly stratified fluid, contained in a closed rectangular cylinder whose bottom is sloped at a small angle α. The full three-dimensionality of this flow leads to effects not evident in axisymmetric containers. We discuss asymptotic results for small Ekman number, E, and Rossby number $\epsilon \ll E^{1/2}$, and a series of experiments conducted at the University of Nottingham. Provided that $\alpha \gg E^{1/2}$, motion occurs on four timescales. The adjustment begins with impulsive motion, caused by the fact that the vertical walls are not streamlines of rigid rotation, and then in times of order α^{-1}, internal waves driven by the sloped bottom wall arise. Ekman-layer eruption on the $E^{-1/2}$ time scale damps this periodic motion, after which the fluid near the lower wall is fully spun up, but the whole of the fluid interior is not. On the E^{-1} time scale, final adjustments occur, to bring the fluid up to the new rotation rate. In the four experiments performed, the rectangular cylinder was $38\,\mathrm{cm} \times 38\,\mathrm{cm}$ in cross-section, with an average depth of $52\,\mathrm{cm}$, and a base slope $\alpha = 10°$. In all cases, the Rossby and Burger numbers were between 0.019-0.022 and 3.1-6.8, respectively, with Ekman number $O\left(10^{-5}\right)$. Throughout each experiment, horizontal-plane velocity components (and corresponding vertical vorticity) were obtained using 2D PIV.

2:23PM D26.00002 Large-scale barotropic circulation in rotating convection , ANTONIO RUBIO, KEITH JULIEN, University of Colorado Boulder, IAN GROOMS, New York University — High resolution DNS for rotating Rayleigh-Benard convection using the non-hydrostatic balanced geostrophic equations (NHBGE) (Julien et al 1998, 2006) was carried out in the geostrophic turbulence regime, revealing the existence of a slow-growing large scale barotropic mode. Such large scale modes have been previously observed as an inverse cascade in stable layer quasi-geostophic dynamics or via instability mechanisms of thermal Rossby waves occuring in presence of sloping endwalls (i.e quasi-geostrophic beta-convection). In this talk we report on a natural large scale organization resulting from smaller scale geostrophic thermal turbulence in an unstably stratified layer. This large scale circulation is discussed in detail in terms of spectra of the term-by-term balances of the baroclinic and barotropic vorticity equations of the NHBGE. Specifically, simulations studies reveal a tendency for the kinetic energy spectrum to follow a -3 scaling exponent at large horizontal scales and a -5/3 exponent in an inertial range of (horizontally) homogenous turbulence occuring at scales smaller than the convective forcing.

2:36PM D26.00003 PIV measurement of flow around an irregularly rotating disk with bluff bodies[1] , SUNGHYUK IM, YOUNG JIN JEON, HYUNG JIN SUNG, KAIST — Flow structure interaction between a rotating disk with bluff bodies and surrounding flow has been evaluated by using a tomographic PIV. To examine the dynamics of disks and the fluid flow simultaneously, fluorescence tracer particles and long pass filters were used. Particles and a marked surface pattern were separated by an image processing. The geometries of the bluff bodies and the disk were then obtained by analyzing the marked pattern. Subsequently, a particle volume was reconstructed by MLOS-SMART. The particle displacement was then calculated by the PIV algorithm from the reconstructed particle volume. Rotating disk dynamics and fluid flow were discussed with variety of bluff body shapes and arrangements. Furthermore, the influence of bluff bodies on the flow field was also considered.

[1]This study was supported by the Creative Research Initiatives of NRF/MEST (No. 2011-0000423) of Korea.

2:49PM D26.00004 Time-Resolved PIV in the Flow Around Cylinders Under the Effect of Coriolis Forces , FILIPPO COLETTI, Stanford University, IGNACIO MAYO YAGUE, TONY ARTS, von Karman Institute for Fluid Dynamics — The effect of Coriolis forces on the dynamics of bluff body wakes is relevant to fields as diverse as turbomachinery and meteorology. Nevertheless such flows are largely unexplored, due to practical difficulties in measuring velocity in the rotating frame of reference. We present both ensemble-averaged and time-resolved PIV measurements around square and circular cylinders at Re=1000, in the presence of system rotation around an axis parallel to the cylinder. The PIV system, including a continuous laser diode and a high speed CMOS camera, is put in rotation on the same turntable as the test section at an angular velocity of 130 rpm, for a rotation number of Ro=0.13. The unique arrangement allows the same level of accuracy and spatio-temporal resolution as in a non-rotating rig. Coriolis forces break the wake symmetry, resulting in one cyclonic and one anti-cyclonic shear layer shed from the opposite sides of the cylinder. For the square cylinder the fluid entrainment from the cyclonic to the anti-cyclonic side deforms the flow topology to the point that only one focus is present in the mean wake. Time series and space-time two-point correlations suggest an overall loss of periodicity and coherence in the presence of system rotation.

3:02PM D26.00005 Bubble deformability is crucial for strong drag reduction in turbulent Taylor-Couette flow[1] , CHAO SUN, DANIELA NAREZO GUZMAN, DENNIS P.M. VAN GILS, DETLEF LOHSE, Physics of Fluids Group, University of Twente — Bubbly Taylor-Couette flow in the turbulent regime is studied both globally and locally at Reynolds numbers of $5.1 \times 10^5 - 2.0 \times 10^6$ for pure inner cylinder rotation. We measure the drag reduction (DR) based on the global torque for global gas volume fractions (α_{global}) up to 4%, and observe a moderate DR for Re $= 5.1 \times 10^5$, and a strong DR for Re $= 1.0 \times 10^6$ and 2.0×10^6. Remarkably, more than 40% of DR is achieved for $\alpha_{global} = 4\%$ at Re $= 2.0 \times 10^6$. We investigate the statistics of the liquid flow velocity, and directly measure the local bubble concentration and Weber number for two Reynolds numbers in different drag reduction regimes, i.e. Re $= 1.0 \times 10^6$ (strong DR) and 5.1×10^5 (moderate DR). By combining global and local measurements we reveal that bubble deformability is crucial for strong drag reduction in bubbly turbulent Taylor-Couette flow.

[1]This work was financially supported by technology foundation STW in The Netherlands.

3:15PM D26.00006 PIV measurements of a jet impinging on an opened rotor-stator system at low gap spacing , THIEN NGUYEN, JULIEN PELLÉ, SOUAD HARMAND, Univ Lille Nord de France, F-59000 Lille, France — The current work experimentally investigates the flow characteristics of an air jet impinging to an opened rotor-stator configuration at a low nondimensional spacing $G = 0.02$ and very low aspect ratio $e/D = 0.25$. The rotational Reynolds numbers varied from 0.33×10^5 to 5.32×10^5 while the jet Reynolds numbers ranged from 17.2×10^3 to 43×10^3. PIV measurements were performed at three axial planes for the entire disk diameter. The obtained PIV results agreed with those obtained by LDA measurements and numerical simulation reported in Poncet et al. 2005 (Physics of Fluids 17, 075110). A recirculation flow region, which centered at the impinging point and possessed high turbulent intensities, was observed. The mean flow and turbulent intensities were evaluated with the local heat transfer coefficients measured by Pellé and Harmand 2009 (Applied Thermal Engineering 29: 1532-1543). It is shown that the local peaks and the gradually rising of the radial heat transfer coefficients Nu are due to the secondary peaks and the increases near the outer radius of the turbulent intensity distributions respectively. POD analysis was applied to the cases of the impinging jet with and without rotation. It is shown that the first POD mode captured nearly 60% total kinetic energy and the low-order POD modes revealed a spiral structure in the jet-dominated region.

3:28PM D26.00007 Blade Vortex Interaction of Bi-directional flow in a Uni-directional Impulse Turbine , CARLOS VELEZ, University of Central Florida — Uni-directional impulse turbines are used for the extraction of wave energy by converting oscillating air flow generated by waves into uni-directional rotational energy. The symmetric airfoil design requires a large camber, in order to function in bi-directional flow, which creates a large boundary layer separation region towards the trailing edge of the blade. A three-dimensional, viscous, transient turbulent CFD model with rotating reference frame is created to model the blade vortex interaction (BVI) which occurs during transient bi-directional air flow. Various LES models are compared to determine an adequate turbulence model to accurately resolve the vortices created on the blade trailing edge. A study of the adverse effects of this BVI is conducted and a novel blade jet technique is introduced to prevent the separation of air flow from the trailing edge of the blade. Results show strong stresses arise from BVI during bi-directional transitional flow and the effectiveness of the blade jet technique in diminishing flow separation is successfully demonstrated. Results indicate that the increase in blade lift is linearly proportional to the blade jet mass flow rate once the jet velocity reaches approximately 120% of the turbine inlet velocity and that the increase in efficiency created by the blade jets are greater than the loss in efficiency in reducing the mass flow rate extracted from the inlet to the blade jet.

3:41PM D26.00008 Validation of a Transient Rotating Reference Frame CFD Model , OTHMANE KHOUNGUI, JUSTIN LADD, CARLOS VELEZ, University of Central Florida — Uni-directional impulse turbines are used for the extraction of wave energy by converting oscillating air flow generated by waves into uni-directional rotational energy. Due to the inconsistent nature of ocean waves, airflow within an OWC is bi-directional and inherently transient. Such complex fluid dynamics require a varying rotor RPM incorporated in the CFD simulation to adequately resolve the flow field during turbine startups and changing air flow direction. The software Numeca is used to introduce a user defined function which defines a varying rotor rpm in a three dimensional transient viscous simulation of air flow through a uni-directional turbine. A scaled turbine prototype is used in a wind tunnel to measure the rotors RPM and Torque. Additionally, a radial pressure profile is developed in front and behind of the rotor blades. The experimental data is used to validate the accuracy of this varying rotating reference frame CFD model.

3:54PM D26.00009 A Parametric Study of Unsteady Rotor-Stator Interaction in a Simplified Francis Turbine[1] , ALEX WOUDEN, JOHN CIMBALA, BRYAN LEWIS, Pennsylvania State University — CFD analysis is becoming a critical stage in the design of hydroturbines. However, its capability to represent unsteady flow interactions between the rotor and stator (which requires a 360-degree, mesh-refined model of the turbine passage) is hindered. For CFD to become a more effective tool in predicting the performance of a hydroturbine, the key interactions between the rotor and stator need to be understood using current numerical methods. As a first step towards evaluating this unsteady behavior without the burden of a computationally expensive domain, the stator and Francis-type rotor blades are reduced to flat plates. Local and global variables are compared using periodic, semi-periodic, and 360-degree geometric models and various turbulence models (k-omega, k-epsilon, and Spalart-Allmaras). The computations take place within the OpenFOAM® environment and utilize a general grid interface (GGI) between the rotor and stator computational domains. The rotor computational domain is capable of dynamic rotation. The results demonstrate some of the strengths and limitations of utilizing CFD for hydroturbine analysis. These case studies will also serve as tutorials to help others learn how to use CFD for turbomachinery.

[1]This research is funded by a grant from the DOE.

4:07PM D26.00010 Stirring in a tilted-rotating tank , THOMAS WARD, DAVID SWAN, ANDREW WHITE, North Carolina State University — Inhomogeneous fluid mixing in a tilted-rotating tank is discussed at O(1-10) Reynolds and small < 1 capillary numbers. At low Reynolds numbers the flow exhibits two large vortices. As the Reynolds number increases to the laminar regime, the two vortices exhibit interactions with the bottom wall and begin a cascading effect that is similar to the well known Moffatt (J. Fluid Mech., 1964) vortices in Stokes flow in cavities. The additional vortices aid in transporting material from the the walls to the bulk of the region between them. But the vortices also intensify in magnitude with increasing rotation rate leading to the appearance of KAM surfaces, which are barriers to efficient mixing. Experiments are performed to study dispersing water in vegetable oil (5% water by volume) and using laser fluorescence to illuminate the vortices via experimental Poincaré mapping. The resulting images are analyzed to determine the mixed cross sectional area versus elapsed time as a function of the system parameters which are the tilt angle and Reynolds number.

Sunday, November 20, 2011 2:10PM - 4:20PM –
Session D27 Focus Session: Wind Energy Fluid Dynamics II Ballroom I-II

2:10PM D27.00001 Competing mechanisms of momentum transport in large wind farms[1] , JOHAN MEYERS, Mechanical Engineering, University of Leuven, CHARLES MENEVEAU, Mechanical Engineering & CEAFM, Johns Hopkins University — In very large wind farms in the atmospheric boundary layer, energy, and momentum are on average transported from layers above the farm downward towards the turbines (Calaf, Meneveau, Meyers, Phys. Fluids 2010). In the current work, we investigate in more detail the three-dimensional flows of mass, momentum and energy towards individual turbines, based on a suite of large-eddy simulations. We find that there are two competing mechanisms which bring momentum to the turbines, i.e. a sideways flux, and a top-down flux of momentum (sideways fluxes themselves are fed by a top-down flux in regions outside the turbine wake area). For large spanwise turbine spacings, sideways momentum fluxes are dominating; for small spanwise spacings, the top-down mechanism is dominant. Inspired by these observations, we propose a new integral model for wind-farm performance, in which competing fluxes of momentum are represented by closed analytical expressions obtained by integrating momentum equations over different regions in the ABL.

[1]The research of CM is supported by NSF AGS 1045189.

2:23PM D27.00002 Large-eddy simulation of atmospheric boundary layer flow through wind farms

, FERNANDO PORTE-AGEL, YU-TING WU, VALERIO IUNGO, Ecole Polytechnique Federale de Lausanne (EPFL), Switzerland, HAO LU, University of Minnesota, USA — Simulating atmospheric boundary layer flow and its interactions with wind turbines is of great importance for optimizing the design (layout) and overall efficiency of wind farms. This presentation focuses on recent efforts to develop and validate a large-eddy simulation (LES) framework for wind energy applications. The subgrid-scale fluxes of momentum and heat are parameterized using tuning-free Lagrangian scale-dependent dynamic models (Stoll and Porte-Agel 2006). The turbine-induced forces are parameterized using two types of models: an actuator disk model that allows for non-uniform distribution of the forces and includes rotational effects (Wu and Porte-Agel 2011); and an actuator line model that distributes the forces on lines that follow the position of the blades. The LES code is validated against wind-tunnel measurements collected with hot-wire anemometry inside and above a large model wind farm. The characteristics of the wind farm wakes simulated with the proposed LES framework, and particularly the spatial distribution of the velocity deficit and turbulence intensity, are in good agreement with the measurements. Finally, in order to extend the LES validation to field conditions, results are presented from a field experiment aiming to characterize wind-turbine wakes using three scanning wind LiDARs.

2:36PM D27.00003 Power measurements on a stratified 3 × 3 wind turbine array

, DOMINIC DELUCIA, NICHOLAS HAMILTON, RAÚL BAYOÁN CAL, Portland State University — Extracting energy out of the wind is the ultimate goal of wind turbine technology and understanding its variation due to the local environment is of interest. Power measurements are carried out in a wind turbine array boundary layer. Measurements are done independently using a torque sensing device which employs strain gages to measure it; following the design of Kang & Meneveau (2010).[1] The power curves are obtained based on a three by three array submerged in an atmospheric turbulent boundary layer, which is generated through the use of roughness elements, an active grid and spires. The power measurements are acquired for the turbines in the array while modifying the local environment. This is done using a thermal floor where an unstable boundary layer is created and then compared with a neutral case.

[1] H. S. Kang and C. Meneveau (2010), Meas. Sci. & Tech. 21, 105206.

2:49PM D27.00004 LES investigation of turbine spacing effects in wind farms[1]

, XIAOLEI YANG, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, Department of Civil Engineering, 2 Third Avenue SE, Minneapolis, MN 55414, USA — We study turbine spacing and layout effects in large wind farms using large-eddy simulation (LES) with the actuator disk model to represent individual turbines. The actuator disk model is implemented in our second-order accurate immersed boundary finite-difference solver using the discrete delta functions to interpolate the velocities on the disk and distribute the body forces to the surrounding fluid points. For aligned wind farms, the effects of the streamwise and spanwise spacings on the extracted power and turbulence intensities in the vicinity of the turbines are systematically studied. A model for the effective roughness length is also developed based on the present numerical results. As part of our future work, the effect of turbine spacing in staggered wind farms will be investigated.

[1] This work was supported by Department of Energy DOE (DE-EE0002980) and Xcel Energy through the Renewable Development Fund (grant RD3-42). Computational resources were provided by the University of Minnesota Supercomputing Institute.

3:02PM D27.00005 Large Eddy Simulation study of scalar transport in fully developed wind-turbine array boundary layers[1]

, MARC CALAF, MARC B. PARLANGE, School of Architecture, Civil and Environmental Engineering, EPFL, Switzerland, CHARLES MENEVEAU, Mechanical Engineering & CEAFM, Johns Hopkins University — Large wind farms are attaining scales at which two-way interactions with the atmospheric boundary layer must be taken into account. A recent study by Baidya et al. (PNAS 2010) has shown that wind farms increase scalar fluxes at the surface. Numerical simulations from Calaf et al. (Pof 2010) together with laboratory experiments from Cal et al. (JSRE 2010) showed that the friction velocity underneath the wind turbines is decreased. Conversely, above the turbine, friction velocity is increased. To shed light onto the relevant phenomena, a suite of Large Eddy Simulations of an infinite (fully developed) wind turbine array boundary layer, including passive scalar transport, is performed. Results clearly show an overall increase of scalar fluxes in the presence of wind turbines, of about 10-15%. And this increase is not highly dependent on wind turbine loading or spacing. This resultant increase in the scalar fluxes can be explained through a balance between two competing effects. Further, following the approach of Calaf et al. (PoF 2010), a single-column model has been developed which confirms the observed trends.

[1] Funding: Swiss Sci. Found. (200021-107910/1) & NSF (AGS-1045189)

3:15PM D27.00006 Turbine layout effects on the flow structure inside an above large wind farms

, LEONARDO CHAMORRO, ROGER ARNDT, FOTIS SOTIROPOULOS, University of Minnesota — An understanding of the role of the wind farm layout on the vertical momentum transport above a wind farm is essential to improve energy production of the different turbines. We investigate the turbulent structure of the flow inside and above a large model wind farm (roughly in fully developed conditions). The large array of turbines consisted of several columns of turbines spaced three abreast in an aligned configuration. The length of the wind farm was over fifteen boundary layer thicknesses. Turbine spacing of 6, 8, 10 and 12 rotor diameter was considered for the analysis. Full characterization of the turbulent flow was obtained between two rows of turbines far inside the wind farms in a vertical plane parallel to the direction of the flow and two spanwise-vertical planes were also included. A cross-wire anemometer was used to obtain high resolution measurements of streamwise and vertical velocity components at various locations at 10 KHz for a sampling period of 120 s at each location. Turbulence statistics, scale-to-scale interaction and TKE budget terms are evaluated to determine the role of the turbine layout on the turbulent dynamics of the flow.

3:28PM D27.00007 Wind Resource Evaluation at the Caltech Field Laboratory for Optimized Wind Energy[1]

, QUINN MULLIGAN, MATTHIAS KINZEL, JOHN DABIRI, Caltech — Wind resources are evaluated at the Caltech Field Laboratory in order to understand how an array of vertical-axis wind turbines extracts energy from the flow. A tower with sonic anemometers placed every meter over the turbine's rotor height is deployed in upwind and downwind positions relative to the array of turbines to obtain the three dimensional wind velocity vectors. Upwind of the array, far enough to be considered free stream, the measured velocity profile represents the turbulent boundary layer flow at the site. Downwind, the measured wind velocities are reduced significantly and display a smaller variance over the rotor height. The topmost sensor, located above the top of the rotor height, reports flow velocities close to the free stream quantities. Sweeps and ejections are both present in the downwind velocity profile. The talk will present the data from these field measurements, discuss the similarities and differences to canopy flows and draw conclusions about the interaction between the wind turbine array and the flow.

[1] The financial support of the Moore Foundation is gratefully acknowledged.

3:41PM D27.00008 Experimental Study of Aligned and Staggered Wind Farms in a Convective Boundary Layer

COREY MARKFORT, WEI ZHANG, Univ of Minnesota, FERNANDO PORTE-AGEL, WIRE-ENAC, EPFL — Wind farm-atmosphere interaction is complicated by turbine configuration and thermal effects on momentum and kinetic energy fluxes. Wind farms of finite length have been modeled as increased surface roughness or as a sparse canopy; however it is not clear which approach is more appropriate. Experiments were conducted in a thermally controlled boundary layer wind tunnel, using a custom x-wire/cold wire and surface heat flux sensors, to understand the effect of aligned versus staggered turbine configurations on momentum absorption and flow adjustment in a convective boundary layer (CBL). Results for experiments of a large farm show the span-wise averaged flow statistics exhibit similar turbulent transport properties to that of canopy flows. The wake adjusts within and grows over the farm more quickly for a staggered compared to an aligned farm. Using canopy flow scaling, we show that the flow equilibrates faster and the overall momentum absorption is higher in a staggered compared to an aligned farm. Wake recovery behind a single turbine is facilitated by buoyancy in a CBL (Zhang et al. under review). We find a similar effect in wind farms resulting in reduced effective roughness and momentum absorption. We also find a reduction of surface heat flux for both wind farms, but greater for the staggered farm.

3:54PM D27.00009 Experimental investigation of turbulence characteristic around a scaled wind turbine[1]

RAMIRO CHAVETZ ALARCON, New Mexico State University, B.J. BALAKUMAR, Los Alamos National Laboratory, FANGJUN SHU, New Mexico State University — Experiments on a scaled model wind turbine, designed using blade element momentum theory, were performed under laminar inflow conditions with and without yaw. A detailed dataset containing wake structure variations under yawed inflow was obtained to provide useful validation data for certain classes of simulation codes. Phase locked PIV experiments performed at various blade orientations (phases) showed that the turbulence characteristics in the mid- to far-wake region is approximately axisymmetric. The power extracted by the model was obtained from the horizontal velocity deficit observed at the wake and compared with the power obtained from torque sensor measurements. Significant differences between these two measurements demonstrate the importance of losses due to viscous and turbulent dissipation.

[1]Supported by Los Alamos National Laboratory's LDRD program through Grant # 20100040DR.

4:07PM D27.00010 Boundary Layer Effects on Wind Turbine's Tip Vortices using PIV Measurements[1]

DAVID GREEN, Dept. of Aerospace Engineering and Mechanics, UMN, LEONARDO CHAMORRO, FOTIS SOTIROPOULOS, ROGER ARNDT, Dept of Civil Engineering, UMN, JIAN SHENG, Dept. of Aerospace Engineering and Mechanics, UMN — Understanding the complex interactions between vortical flow structures of the Horizontal Axis Wind Turbine (HAWT) and the atmospheric boundary layer is crucial to optimize blade design and turbine spacing in a wind farm. Tip vortices shed by the blades often play a key role. This paper focuses on the boundary layer flow interacting with a single turbine and multiple turbines in an in-line configuration using Particle Image Velocimetry (PIV) technique. The model has three blades with a span of 6.4 cm and 1.5 cm chord length. The tip speed ratio is set at roughly 5. The models are roughly within one fourth of the boundary layer. PIV measurement is phase locked on the position as the blade is passing through the measurement plane. Flow fields are captured up to 10 diameters or 87 chord lengths downstream. The effects on the turbine-generated vortical structures in a turbulent boundary layer are analyzed. Comparison to tip vortices produced in a free-stream mean flow will also be presented.

[1]Funded by NSF and DoE

Sunday, November 20, 2011 4:40PM - 6:50PM –

Session E1 Geophysical Flows: Oceanography I 301

4:40PM E1.00001 Shear Instabilities, Internal Waves and Turbulence in an Equatorial Undercurrent Model

HIEU PHAM, SUTANU SARKAR, KRAIG WINTERS, University of California, San Diego — Direct Numerical Simulation is used to investigate the role of both shear instabilities and internal waves in turbulent mixing in an Equatorial Undercurrent model. The flow condition corresponds to a surface mixed layer moving in an opposite direction of an underlying linearly-stratified stream where the gradient Richardson number is larger than 0.25. Holmboe shear instability emerges at the base of the mixed layer, moves at the speed of the local velocity, and ejects wisps of fluid from the bottom stream upward. At the crests of the primary Holmboe instability, a secondary Kelvin-Helmholtz shear instability causes isopycnal overturns resulting in significant turbulent mixing. Vortices formed by the Kelvin-Helmholtz instability are occasionally ejected downward and stretched by the local shear into horseshoe vortices creating intermittent bursts of turbulence in the bottom stream. Horizontally propagating internal waves with wavelength and frequency equal to that of the Holmboe instability are trapped in the bottom stream where they persist longer than the turbulent mixing caused by either the two shear instabilities or the horseshoe vortices. Internal waves and turbulence in the bottom stream of our model have characteristics similar to the near-N oscillations and the deep-cycle turbulence observed in the Pacific Equatorial Undercurrent.

4:53PM E1.00002 ABSTRACT WITHDRAWN –

5:06PM E1.00003 Excitation of harmonic modes by an internal wave beam incident on a simulated ocean pycnocline[1]

SCOTT WUNSCH, ALAN BRANDT, Johns Hopkins University — Laboratory experiments have been performed to investigate the reflection of an internal wave beam with a "pycnocline" layer situated below an unstratified layer in order to simulate observed oceanic processes. An oscillating cylinder was used to generate internal wave beams in the well-known "St. Andrew's Cross" pattern. Interactions with the pycnocline were observed using the synthetic schlieren technique. As the beam refracted into the pycnocline, a discrete spectrum of harmonic modes was excited. For modest pycnocline stratifications (relative to the stratified layer below) only a few modes were observed, containing only a few percent of the kinetic energy of the primary beam. However, strongly stratified pycnoclines resulted in many excited modes and a more significant transfer of energy. The observed nonlinear energy transfer may be indicative of the validity of a postulated mechanism for the formation of oceanic internal solitary waves by internal waves incident on the pycnocline.

[1]Supported by Office of Naval Research (N00014-08-1-0290), Dr. Ron Joslin.

5:19PM E1.00004 Nonlinear Harmonic Generation During the Interaction of an Internal Wave Beam with a Sharp Oceanic Pycnocline

PETER DIAMESSIS, PAUL RICHTER, Cornell University, SCOTT WUNSCH, Johns Hopkins University — The phenomenon of nonlinear harmonic generation upon the encounter of an internal wave beam with a sharp oceanic pycnocline underlying a well-mixed layer is examined using 2-D direct numerical simulation. Use of a spectral multidomain penalty scheme enables the detailed resolution of phenomena within the pycnocline. Values of the ratio, r, of pycnocline to lower layer Brunt-Vaisala (BV) frequency and pycnocline thickness, h/λ_x, normalized by the incident horizontal wavelength are considered in the ranges [1,10] and [0.1,1], respectively. Nonlinear harmonic generation is enhanced with increasing values of r/h, i.e. the gradient of the BV frequency within the pycnocline. We provide scaling laws for the content of the first two harmonics as a function of environmental parameters and explore the possibility of a non-monotonic dependence which indicates particular regimes for resonance.

5:32PM E1.00005 Interaction of an internal tide beam with an upper ocean pycnocline , BISHAKHDATTA GAYEN, SUTANU SARKAR, UCSD — Direct numerical simulation and linear inviscid theory are used to study the interaction of an internal wave beam with an upper ocean pycnocline at different values of Froude number, Fr. At low Fr, both theory and numerics agree well as to the behavior of the internal wave beam after reflection. At moderate values of Fr, nonlinear response initiates the formation of higher harmonics. The harmonics that have frequency higher than the buoyancy frequency of the lower medium, are unable to propagate into the lower medium. These harmonics, affected by multiple reflections inside the pycnocline, are trapped. At high value of Fr, the IW beam undergoes shear instabilities during its tunneling through the transitional layer of the pycnocline. The role of turbulence in modifying the internal wave beam structure will be discussed.

5:45PM E1.00006 On the onset of instability in self-induced shears by large amplitude internal waves[1] , ROBERTO CAMASSA, CLAUDIO VIOTTI, University of North Carolina, UNC RTG FLUIDS GROUP TEAM — Large amplitude internal waves in stratified fluids generate a shear layer which can support Kelvin-Helmholtz instabilities when the pycnocline is sharp enough. The way perturbations propagate along such nonparallel shear flows and eventually excite the local unstable modes is not fully understood, and differences from the parallel setup can be expected. Here we consider the specific case of a solitary wave interacting with prescribed upstream baroclinic perturbations with the aim of assessing the role of the self-induced shear flow on the evolution of the perturbation. We present a few preliminary results documenting how internal waves exhibit a subtle selectivity on the nature of the perturbation itself, which affects the outcome between the two extremes from net amplification to net damping, even in locally-unstable conditions. By varying the parameters of the perturbation (wavenumber, phase speed or frequency) we find that frequency correlates best with the overall growth/decrease trend, thereby favoring an interpretation of the underlying mechanism in terms of quasi-spatial instability.

[1]NSF DMS RTG 0943851, NSF DMS 1009750

5:58PM E1.00007 Oblique Collisions of Internal Gravity Wave Beams[1] , T.R. AKYLAS, HUSSAIN KARIMI, MIT — Nonlinear interactions between two colliding internal gravity wave beams in a stratified fluid are studied theoretically, making use of small-amplitude expansions. Such collisions can give rise to secondary wave beams with frequencies equal to the sum and difference of the frequencies of the primay beams, a mechanism that is believed to contribute to the energy cascade from the tides to ocean mixing. Earlier work has been confined to the special case when the propagation directions of the primary beams, and hence the induced higher-order beams, lie in the same vertical plane. Here, the general three-dimensional (3D) configuration where the colliding beams approach each other obliquely, is considered. Based on suitable radiation conditions, the wave characteristics and direction of secondary beams are deduced, thus generalizing the known selection rules for plane collisions to the 3D case. Moreover, explicit expressions for the induced-beam profiles are derived and their dependence on obliqueness of collision is examined.

[1]Supported by NSF

6:11PM E1.00008 Resonant Triad Instability in Stratified Fluid , SYLVAIN JOUBAUD, Laboratoire de Physique-ENS de Lyon, JAMES MUNROE, Department of Physics and Physical Oceanography, PHILIPPE ODIER, THIERRY DAUXOIS, Laboratoire de Physique-ENS de Lyon — Internal waves are believed to be of primary importance as they affect ocean mixing and energy transport. Several processes can lead to the breaking of internal waves and they usually involve non linear interactions between waves. In this work, we experimentally study the resonant triad instability (also called Parametric Subharmonic Instability), which provides an efficient way to transfer energy from large to smaller scales. It corresponds to the destabilization of a parent wave and the spontaneous emission of two daughter waves, of different frequencies and wave numbers. We observe the experimental conditions under which a monochromatic vertical mode-1 wave is unstable. Using a time-frequency analysis, we are able to follow the evolution of the instability. We quantitatively measure the growth rate of the amplitude of the two daughter waves and compare with theoretical predictions.

6:24PM E1.00009 Subsurface Signature of a Reflecting Internal Wave Beam , QI ZHOU, PETER DIAMESSIS, Cornell University — We examine the subsurface signature of an internal wave beam reflecting off a free-slip slip top surface in a linearly stratified water column using 3-D direct numerical simulations. A spectral multidomain penalty scheme in the vertical enables finer resolution of the subsurface region. The wave beam is modelled based on a recently obtained characterization of the internal wave field radiated by a stratified turbulent wake of a towed sphere. Subsurface current and strain fields are quantified as a function of beam geometry, steepness and phase-line-tilt angle with respect to the vertical. We also compute the induced surface displacements and discuss the potential of nonlinear harmonic generation.

6:37PM E1.00010 Reflection of internal gravity waves at a bottom topography with near-critical slope , VAMSI KRISHNA CHALAMALLA, BISHAKHDATTA GAYEN, SUTANU SAKAR, UCSD, ALBERTO SCOTTI, UNC — Direct numerical simulation is performed to study plane internal wave reflection at a sloping bottom at different values of Froude number, Fr. The slope angle is also varied in a range of near-critical values. At low Fr, the numerical results agree well with linear inviscid theory of near-critical internal wave reflection. With increasing Froude number, the reflection process becomes increasingly nonlinear with the formation of higher harmonics and subsequently fine scale turbulence. At a critical value of Fr, turbulence is initiated via convective instability. Also, turbulent intensities are more pronounced for somewhat off-critical reflection compared to exactly critical reflection. As the Froude number increases, the near wall shear plays a dominant role in critical reflection by enhancing turbulence compared to off-critical reflection. For a fixed slope angle, as the Froude number increases the fraction of the input energy converted into the turbulent kinetic energy increases and saturates at higher Froude numbers.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E2 Instability in Boundary Layers I 302

4:40PM E2.00001 Boundary-layer instability & transition on a flared cone in a Mach 6 quiet wind tunnel[1] , JERROD HOFFERTH, WILLIAM SARIC, Texas A&M University — Measurements of boundary-layer transition location and instability growth on a sharp-tipped 5°-half-angle flared cone were conducted in a low-disturbance Mach 6 wind tunnel at a freestream unit Reynolds number of 10×10^6/m. Under quiet flow at these conditions, the boundary layer becomes transitional near the base of the cone, where significant second-mode instability growth is evident. Transition location is determined using an array of embedded thermocouples, and instability development is observed in mean and fluctuating mass flux data using hotwire anemometry. The present work seeks to reproduce and build upon previous experiments which used the same test article and similar diagnostics in the facility's former installation at NASA Langley. Together with comprehensive measurements of the freestream disturbance environment, these baseline cone data characterize the facility's performance relative to that in its previous installation. In addition, the current campaign establishes experimental readiness for future research, which will study the effects of periodic surface roughness and controlled-input disturbances.

[1]AFOSR/NASA National Center for Hypersonic Research in Laminar-Turbulent Transition; Grant FA9550-09-1-0341

4:53PM E2.00002 Plasma Roughness for Transition Control in a 3-D Supersonic Boundary Layer[1]

, CHAN-YONG SCHUELE, ERIC MATLIS, THOMAS CORKE, University of Notre Dame, STEPHEN WILKINSON, NASA Langley Research Center — The design and use of patterned "plasma roughness" for control of transition to turbulence of the boundary layer with a supersonic free-stream is presented. The plasma roughness consisted of an azimuthal array of 20 nm thick electrodes that were equally spaced around the cone tip, just upstream of Branch I for cross-flow instability growth. The electrodes were part of a DBD arrangement that produced an azimuthally periodic stationary body force that acted on the flow. The azimuthal spacing of the electrodes was designed to either enhance the most amplified stationary mode growth ($m = 45$ in this case), or to excite a sub-critical mode number ($m = 68$) that was designed to suppress the most amplified mode. The experiment was performed on a 14° right-circular cone placed at a 4.3° angle of attack in the NASA LaRC SLDT. Measurements consisted of azimuthal profiles of the total pressure just above the cone surface. These documented the mean flow distortion produced by the growing stationary cross-flow modes. Comparisons were made with and without the plasma roughness, as well as against passive patterned roughness with the same azimuthal mode numbers. The results indicated that the stationary cross-flow modes were receptive to the patterned plasma roughness, and that Re_{trans} was increased.

[1]Supported under NASA Cooperative Agreement NNX08AB22A

5:06PM E2.00003 Nonlinear wavepackets in unstable boundary layers: experiments and modeling[1]

, MARCELLO MEDEIROS, University of Sao Paulo — The work involved wind tunnel experiments and weakly nonlinear modeling of modulated TS waves. It covered from wavetrains emanating from a harmonic point source to the wavepackets from pulses. The nonlinear regime of these waves exhibit three stages. The first one was governed by direct nonlinear forcing and produced longitudinal streaks. The second was governed by K- type instability. Both stages were triggered by the spanwise modulation. The third nonlinear stage was governed by subharmonic, H - type instability and was exclusive of streamwise modulated waves. The phase of the waves relative to the modulation envelope was also found to be very important for the third stage only. In general, it was found that the modulation did not affect the kernel of the secondary instabilities observed. The effect of both streamwise and spanwise modulation was on the deterministic production of seeds for these instabilities, via quadratic interaction of the existing modes. The weakly nonlinear model explained many features that at first seemed surprising. For instance, the effect of phase on the subharmonic stage was explained by the production of subharmonic seeds from the direct nonlinear interaction of both the primary modes and the fundamental instability modes.

[1]Support was received from AFOSR/USA and CNPq/Brazil.

5:19PM E2.00004 The Effect of Moderate Freestream Turbulence on Crossflow Instability and Transition[1]

, ROBERT DOWNS, EDWARD WHITE, Texas A&M University — Previous research indicates that the generation of stationary and traveling crossflow modes is dependent on the level of freestream turbulence, *Tu*. Recent experiments in low turbulence environments have also shown that the initial amplitudes of stationary crossflow disturbances arising from surface roughness are influenced by small variations in *Tu*. Preliminary results are presented concerning experimental investigation into the role of moderate levels of freestream turbulence ($Tu = 0.025\%$ - 0.2%) in boundary layer receptivity to surface roughness. Transition induced by crossflow instability is studied using a 45-degree swept wing model in the low turbulence Klebanoff-Saric Wind Tunnel. Naphthalene flow visualization is used to assess transition location and detailed hotwire anemometry measurements are made to quantify these initial disturbance amplitudes.

[1]Supported by AFOSR under grant FA9550-07-1-0312

5:32PM E2.00005 Natural by-pass boundary layer transition[1]

, SHAHAB SHAHINFAR, JENS H.M. FRANSSON — The present measurement campaign on the free-stream turbulence induced boundary layer transition scenario has provided a unique set of experimental data, with potential to enhance the understanding of the effect of the free-stream turbulence characteristic length scales on the transition location and not only the turbulence intensity, which has been the focus in most previous studies. Recent investigations where the turbulence intensity has been kept essentially constant, while the integral length scale has been changed, show that the transition location is advanced for increasing length scale. However, the present data show that the integral length scale has a relatively small influence on the transition location as compared to the turbulence intensity and data analyses are now directed towards enhanced understanding of how the different parts of the incoming energy spectrum affects the energy growth inside the boundary layer.

[1]The Linné Flow Centre is greatly acknowledged for financial support of this work. The APS 2011 attendance is financed through the AFRODITE programme.

5:45PM E2.00006 An experimental study of the edge effect on transition of the rotating-disk boundary-layer flow

, SHINTARO IMAYAMA, R.J. LINGWOOD, P. HENRIK ALFREDSSON, Linne FLOW Centre, KTH Mechanics, Royal Institute of Technology, Stockholm, Sweden — Lingwood [J. Fluid Mech., 299, 17 (1995)] showed that the flow instability in the rotating-disk boundary layer is not only of convective nature but also that the flow becomes absolutely unstable. Furthermore, in the absence of bypass mechanisms, the absolute instability triggers nonlinearity and transition to turbulence at a fixed Reynolds number (Re). Healey [J. Fluid Mech., 663, 148 (2010)] suggested that the observed spread (albeit small) in transition Re in different experiments is an effect of the Re at the disk edge and provided a nonlinear model to take this effect into account. Here, we further investigate this problem experimentally with hot-wire measurements on a rotating polished glass disk with a diameter of 474mm and a total imbalance and surface roughness less than 10μm. To investigate the influence of the disk edge, we vary Re at the disk edge by changing the rotational speed and map the development of the disturbance velocity in the radial direction. Furthermore, the effect of a stationary annular plate around the edge of the rotating disk is also investigated. Our experiments show no effect of the disk edge Re on the stability and transition, however there was a shift of both the growth curve and the transition Re by about 10 units with and without the outer stationary plate, with the lower Re observed with the plate.

5:58PM E2.00007 Surface acoustic wave induced flow in micro channels: peculiar flow instability observed in experiment and advances in theory

, OFER MANOR, JAMES FRIEND, LESLIE YEO, Micro/Nanophysics Research Laboratory, Monash University, Clayton, VIC, MICRO/NANOPHYSICS RESEARCH LABORATORY, MONASH UNIVERSITY TEAM — Challenges in microfluidics have initiated a renewed interest in surface acoustic waves (SAW) and in particular Rayleigh SAW for generating high fluid velocities and mixing in microchannels. SAW are generated by piezoelectric actuators that transfer electric to kinetic energy and give rise to different types of flow regimes – Eckart, Rayleigh and Schlichting streaming. Here we extend the theory for the Schlichting boundary layer. We show that due to the complex nature of the Rayleigh SAW, the leading order result for the steady flow in the Schlichting layer is of greater magnitude than previously considered in studies on stationary planar acoustic waves. We confirm this result by measuring the behavior of a dilute suspension of particles in 80 micron thick PDMS channels. In the presence of only a stationary planar acoustic wave in the liquid we observe the weak accumulation of particles in the vicinity of acoustic nodal lines. These structures are easily interrupted by bulk flow. In the presence of Rayleigh SAW, however, particle structures are found to maintain much better integrity and to occupy much larger area under dispersive flow effects.

6:11PM E2.00008 Squire's transformation and 3D Optimal Perturbations in Bounded Parallel Shear Flows , JEAN-MARC CHOMAZ, J. JOHN SOUNDAR JEROME, LadHyX, CNRS-Ecole Polytechnique — The aim of this short communication is to present the implication of Squire's transformation on the optimal transient growth of arbitrary 3D disturbances in parallel shear flow bounded in the cross-stream direction. To our best knowledge this simple property has never been discussed before. In particular it allows to express the long-time optimal growth for perturbations of arbitrary wavenumbers as the product of the gains from the 2D optimal at a lower Reynolds number itself due to the Orr-mechanism by a term that may be identified as due to the lift-up mechanism. This property predict scalings for the 3D optimal perturbation well verified by direct computation. It may be extended to take into account buoyancy effect.

6:24PM E2.00009 The role of the frozen relative Mach number on the stability of boundary layers in chemical non-equilibrium[1] , JILL KLENTZMAN, ERMAN ULKER, ANATOLI TUMIN, University of Arizona — The stability of boundary layers in chemical non-equilibrium is investigated in the inviscid limit. The flow of binary mixtures of oxygen and nitrogen over a flat plate is considered and a parametric study varying the edge temperature and pressure while keeping the frozen edge Mach number constant is conducted. In contract to the case of a calorically perfect gas, in which little to no effect on the stability would be expected, varying these parameters in the case of a gas in chemical non-equilibrium produces significant changes in the stability results. Both cold and adiabatic, non-catalytic walls are considered and the results compared. It is found that wall cooling leads to an increase in the maximum growth rate of the second mode instability and a shift in the second mode to higher wave numbers. The trends observed due to real gas effects reveal that the frozen relative Mach number may play a significant role similar to what has been found in perfect gas flows.

[1]This work was supported by the Air Force Office of Scientific Research (AFOSR)/NASA/National Center for Hypersonic Research in Laminar-Turbulent Transition.

6:37PM E2.00010 The effect of chemistry and transport models on the inviscid stability of boundary layers in binary mixtures of oxygen and nitrogen[1] , ERMAN ULKER, JILL KLENTZMAN, ANATOLI TUMIN, University of Arizona — An inviscid stability analysis of boundary layers in binary mixtures of oxygen and nitrogen in chemical non-equilibrium is conducted. Cold and adiabatic, non-catalytic walls are considered and two different chemistry and transport models are used: the reaction rates and transport models of Gupta et al. (1990) as well as Blottner's transport model (1971) with Park's reaction rates (1989). The results show that changing the chemistry and transport models has a major impact on the boundary layer profiles and the perturbation growth rates when dissociation is significant in the boundary layer. In the model of Gupta et al., the reaction rates do not change with respect to the collision partner in binary mixtures of oxygen, which is in contrast to Park's reaction rates. For binary mixtures of nitrogen, the reaction rates change according to the collision partner for both models. The results of the comparison of different chemistry and transport models show that the flow in binary mixtures of oxygen is more affected by the choice of the model than in binary mixtures of nitrogen.

[1]This work was supported by the AFOSR/NASA/National Center for Hypersonic Research in Laminar-Turbulent Transition.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E3 Waves I 303

4:40PM E3.00001 A wave vortex force formalism for wave-current interaction in strongly sheared flows , ZHIFEI DONG, JAMES T. KIRBY, University of Delaware — In the river plume, waves can be strongly affected by spatial and temporal variations in ambient current fields, which can induce shoaling, refractive and focussing/defocussing and breaking effects in analogy to variations in bathymetry. In turn, waves can modify the current distribution, determining the shape of river plume by affecting its centerline velocity, lateral spreading and plume thickness. As part of the ongoing study of highly concentrated sediment transport in plumes developed by small mountainous rivers, we formulate a general framework for describing the interaction of small amplitude surface gravity waves and relatively strongly sheared currents in finite depth water, where shear can exist in both the vertical and horizontal. In contrast to existing formulations, where the wave at leading order responds to a depth-uniform current field, the present formulation allows for an arbitrary degree of vertical shear, leading to a description of the vertical structure of waves in terms of solutions to a Rayleigh stability equation. The resulting formulation leads to a conservation law for wave action, and forcing terms for the description of the mean flow formulated using the Craik-Leibovich vortex formalism. A special case for strong current with linear shear is discussed. Results are provided to compare with the existing formulations. Applications to numerical model is left for further development.

4:53PM E3.00002 Shallow Water Model Using Adaptive Wavelet Collocation Method[1] , SHANON RECKINGER, Fairfield University, OLEG V. VASILYEV, BAYLOR FOX-KEMPER, University of Colorado at Boulder — The adaptive wavelet collocation method is applied to the shallow water model. This method solves the equations on temporally and spatially varying meshes, which allows higher effective resolution to be obtained with less computational cost. The grid adaptation is achieved by using the ability of wavelet multiresolution analysis to identify and isolate localized dynamically dominant flow structures, e.g., vortices, and to track these structures on adaptive computational meshes. In addition to studying how the shallow water model behaves on non-uniform, time varying grids, this work also sets out to improve the representation of continental topology through an extension of the Brinkman penalization method. This numerical technique works by altering the governing equations in such a way that no slip boundary conditions are enforced. When coupled with the adaptive wavelet collocation method, the flow near a complex boundary can be well defined. In addition the bathymetry is represented in wavelet compressed form, thus allowing active control of the roughness, length scales, etc., plus efficient representation of the detailed bathymetry, with automatic refinement in regions of active interaction of bathymetry and flow structures. The applications presented here include wind-driven flow in a square basin, North Atlantic circulation, and a tsunami simulation.

[1]This work was supported by DOE-CCPP (DE-FG02-07ER64468).

5:06PM E3.00003 A new triad resonance in two-layer density stratified fluids , MOHAMMAD-REZA ALAM, Department of Mechanical Engineering, University of California, Berkeley — In a two-layer density stratified fluid it is known, due to Ball (1964), that two oppositely traveling surface waves may form a triad resonance with an interfacial wave. In the case of a real ocean with a relatively weak stratification, the two surface waves have close wavelengths and the resonant interfacial wave has the wavelength of about half of surface waves. Ball (1964) claims "there are no other interactions" between two surface waves and one interfacial wave. Contrary to this, here we present a new class of triad resonance that obtains between two co-propagating surface waves (with close wavelengths) and a much longer interfacial wave. We present, via theoretical analysis and direct simulation, that for weak stratifications this new class of resonance results in a cascade of (near) resonance interaction that spreads the energy of initial waves to a number of lower and higher frequency waves. The resonance discussed here is in fact more likely to affect the evolution of a spectrum because waves within a typical spectrum are usually co-propagating than oppositely traveling. The significance of the resonance studied here is, particularly, more highlighted in the littoral zones, where the spectrum refracts toward a uni-directional wave train.

5:19PM E3.00004 Numerical study of non-breaking and breaking surface waves over viscous mud flow , YI HU, Tsinghua University, XIN GUO, YI LIU, LIAN SHEN, ROBERT A. DALRYMPLE, Johns Hopkins University — It is well known that water surface waves can be drastically damped over a muddy seabed. To understand the mechanism of wave-mud interaction, we perform DNS of the NS equations for wave propagation over mud, which is modeled as Newtonian fluid with larger density and much higher viscosity than water. A level set method is used to capture water surface and water-mud interface. From the simulations, the velocity and vorticity and the energy budget terms in water and mud are analyzed. The energy flux from water to mud and the dissipation in mud are found to play an important role in the energy budget. For non-breaking waves, despite the wave nonlinearity, the wave dissipation rate is found to be comparable to the predictions of existing theories. For breaking waves, before the impingement of plunging jet on the wave surface and then after about 2.5 wave periods when most of the wave energy is lost, dissipation in mud dominates that in water. During the wave breaking itself, the dissipation rate in water increases sharply and exceeds that in mud. In the presence of mud, the intensity of breaking is reduced compared with the non-mud case.

5:32PM E3.00005 Temporal decay of water waves over a mud layer in a small tank , YOUNES NOURI, ROBERT A. DALRYMPLE, KHATOON MELICK, Johns Hopkins University — Several field observations have recorded drastic dissipation of water waves traveling over a soft muddy bottom. However, current understanding of mechanisms of interaction of waves and mud layer is still at a preliminary level. Here, the temporal decay in amplitude of standing water waves over a layer of kaolinite mud in a small tank was examined to study role of mud layer thickness, density and rheological properties, as well as initial wave amplitude. An important parameter was found to be the history of the mud prior to the tests. The data set provides a suitable benchmark for verification of theoretical and numerical models of waves-mud interaction.

5:45PM E3.00006 Numerical Study of turbulent coherent structures and bubble entrainment under surfzone breaking waves , GANGFENG MA, JAMES KIRBY, FENGYAN SHI, University of Delaware — Wave breaking in the surf zone entrains large volumes of bubbles into the water column. These bubbles are involved in intense interactions with mean flow and turbulence, producing a complex two phase bubbly flow field. It is necessary to describe the dynamics of breaking waves as two-phase flow with air bubbles of appropriate size distribution. The present study employ a three- dimensional large eddy simulation coupled with a two-phase bubbly flow model to investigate the detailed interactions between coherent structures and bubble entrainment in a laboratory scale breaking wave. Our results show that the vortical structures under the wave front are characterized by counter-rotating vortices. These vortices are initially carried downward by the downbursts, and then subject to stretching and bending to form obliquely descending eddies. To study the effects of coherent structures on turbulence and momentum transport, we conduct a TKE budget analysis. It is found that the TKE and Reynolds stress transport are closely related to obliquely descending eddies behind wave crest. Using the two-phase bubbly flow model, we are able to simulate the bubble entrainment and transport after wave breaking. Our results support the conclusion that the obliquely descending eddies have great effects on bubble entrainment and downward dispersion.

5:58PM E3.00007 Large-wave simulation of spilling breaking and undertow current over constant slope beach , ATHANASSIOS DIMAS, GERASIMOS KOLOKYTHAS, AGGELOS DIMAKOPOULOS, University of Patras — The three-dimensional, free-surface flow, developing by the propagation of nonlinear breaking waves over a constant slope bed, is numerically simulated. The main objective is to investigate the effect of spilling breaking on the characteristics of the induced undertow current by performing large-wave simulations (LWS) based on the numerical solution of the Navier-Stokes equations subject to the fully nonlinear free-surface boundary conditions and the appropriate bottom, inflow and outflow boundary conditions. The equations are properly transformed so that the computational domain becomes time-independent. In the present study, the case of incoming waves with wavelength to inflow depth ratio $\lambda/d \approx 6.6$ and wave steepness $H/\lambda \approx 0.025$, over bed of slope $\tan\beta = 1/35$, is investigated. The LWS predicts satisfactorily breaking parameters - height and depth - and wave dissipation in the surf zone, in comparison to experimental data. In the corresponding LES, breaking height and depth are smaller and wave dissipation in the surf zone is weaker. For the undertow current, it is found that it is induced by the breaking process at the free surface, while its strength is controlled by the bed shear stress. Finally, the amplitude of the bed shear stress increases substantially in the breaking zone, becoming up to six times larger than the respective amplitude at the outer region.

6:11PM E3.00008 An Experimental Study of Spilling Breakers in the Presence of Wind and Surfactants[1] , X. LIU, D. WANG, J. H. DUNCAN, University of Maryland — Spilling breaking waves in the presence of light-wind and surfactants are studied experimentally in a wind-wave tank. The breaking waves are mechanically generated with a single wave maker motion that produces a weak spilling breaker in clean water without wind. The crest-profiles of the waves along the center plane of the tank are measured with a cinematic laser-induced fluorescence (LIF) technique. It is found that with a wind speed lower than 2.3 m/s, the wave breaking is initiated with the formation of a bulge-capillary-wave pattern on the forward face of the wave crest. This pattern is dominated by surface tension and is qualitatively similar to the pattern found without wind and surfactants, Duncan et al. (1999). The slope of the back face of the wave crest decreases rapidly with increasing wind speed and the geometrical parameters describing the bulge are found to be linearly proportional to the surface wind drift speed. The physics of the effect of surfactants on these parameters is explained by a longitudinal wave theory.

[1]Supported by the National Science Foundation, Division of Ocean Sciences, Grant OCE0751853.

6:24PM E3.00009 Evolution of deep-water waves under wind forcing and wave breaking effects: Numerical simulations and experimental assessment , ZHIGANG TIAN, Division of Ocean Systems Engineering, Korea Advanced Institute of Science and Technology (KAIST), Daejeon 305-701 Republic of Korea, WOOYOUNG CHOI, Department of Mathematical Sciences, New Jersey Institute of Technology, Newark, NJ 07102 USA — The performance of wave breaking and wind forcing models on the evolution of deep-water waves is evaluated with laboratory experiments. In the experiments, non-breaking and breaking wave groups are generated by the dispersive focusing technique and different wind conditions are considered. Surface elevations are measured with high-speed imaging. For numerical studies, an eddy viscosity model is employed to simulate energy dissipation due to wave breaking. Wind forcing is modeled by introducing a pressure distribution over the water surface in the dynamic boundary condition. The models are incorporated into a wave evolution model, which is solved numerically using a pseudo-spectrum method for the evolution of the wind forced breaking wave groups. Comparisons between the experimental and numerical results and discussions of the numerical model performance will be presented.

6:37PM E3.00010 Statistics of unidirectional random breaking water-waves , LEV SHEMER, Tel-Aviv University, ANNA SERGEEVA, Institute of Applied Physics, RAS — Quasi-random wave groups were studied experimentally in a 300 m long Large Wave Chanel in Hannover. Multiple realizations of several spectral shapes each having random phases of individual harmonics were excited by a computer-controlled wavemaker. Wave field evolution along the tank was recorded by 28 wave gauges and the variation of waves' statistical parameters with the distance from the wavemaker was analyzed. An attempt was made to identify individual breaking events based on the spectrum variation between consecutive wave gauges. It was concluded that energy decay in the high frequency part of the spectrum can serve as a reliable criterion for breaking localization in each realization. The data processing based on the adopted criterion resulted in constructing separate ensembles of events with and without breaking. Statistical processing of those ensembles enabled to assess the effect of breaking on such wave field characteristics as probability of appearance of extremely steep (rogue, or freak) waves, as well on skewness and kurtosis.

Sunday, November 20, 2011 4:40PM - 6:37PM –
Session E4 Drops III: Experiments 307

4:40PM E4.00001 Dynamics of a Disturbed Sessile Drop measured by AFM[1] , PATRICIA MCGUIGGAN, DANIEL GRAVE, Johns Hopkins University, JAY WALLACE, MACS Consulting, SHENGFENG CHENG, Sandia National Laboratories, ANDREA PROSPERETTI, MARK ROBBINS, Johns Hopkins University — A new method for studying the dynamics of a sessile drop by atomic force microscopy (AFM) is demonstrated. A hydrophobic microsphere (radius, r =20 micrometers) is brought into contact with a small sessile drop resting on a hydrophobic surface. When the microsphere touches the liquid surface, the meniscus rises onto the microsphere due to capillary forces. Although the microsphere volume is six orders of magnitude smaller than the drop, it excites the lowest normal resonance modes of the liquid interface. The sphere is pinned at the interface whose small (less than 100 nm) oscillations are readily measured with the AFM. Resonance oscillation frequencies were measured for drop volumes between 5 microL and 200 microL. The results for the two lowest normal modes are quantitatively consistent with continuum calculations for resonance of hemispherical drops with no adjustable parameters. The method may enable sensitive measurements of volume, surface tension and viscosity of small drops.

[1]Supported by the 3M non-tenured Faculty Grant and the National Science Foundation under Grant No. 0709187.

4:53PM E4.00002 Investigation of drop motion through circular orifices[1] , ANKUR DEEP BORDOLOI, ELLEN K. LONGMIRE, Aerospace Engineering & Mechanics, University of Minnesota, XIANGZHAO KONG, MARTIN O. SAAR, Earth Sciences, University of Minnesota — The motion of drops though porous media occurs in numerous science and engineering fields including multiphase fluid flow in the subsurface during groundwater flow, geothermal energy recovery, and geologic carbon dioxide sequestration. Here, we simplify the porous medium to a thin plate with an orifice to study the interactions between the drop and the solid medium. Drops of water/glycerin with diameter, D, are released in a tank of silicone oil with matched refractive index and allowed to fall downward by gravity. After reaching terminal speed, the drops encounter a thin plate with orifice diameter, d, placed horizontally within the surrounding tank. Drop deformation, contact with the orifice, and breakage are investigated using high-speed imaging, and velocity fields are determined by particle image velocimetry (PIV). Effects of diameter ratio d/D, drop Reynolds number, and drop offset with respect to the orifice center are examined. The experimental results are compared to results from numerical simulations using an immiscible, two-color BGK lattice-Boltzmann method performed under similar test conditions.

[1]Supported by DOE (DOE EERE-PMC-10EE0002764).

5:06PM E4.00003 Experimental Studies of Liquid/Liquid Droplets Transport in Curved Microchannels , ZEYAD ALMUTAIRI, University of Waterloo/King Saud University, TOMASZ GLAWDEL, CAROLYN REN, DAVID JOHNSON, University of Waterloo — This study utilizes optical visualization techniques to experimentally investigate the changes in transport properties of liquid droplets while they travel through microchannel turns. This study was prompted by observations that showed the length and speed of the droplet dynamically change while it is travelling through the turn. To gain insightful understanding of this phenomenon, systematic studies were carried out experimentally where the channel width, height, and turn radius were varied together with flow conditions in the low Capillary and Reynolds number flow regimes. Experimental findings reveal that the dynamic changes in droplet transport through curved microchannels are mainly caused by geometrical properties of the turn such as radius to channel width ratio and the height ratio. We also believe that the behavior of droplets in turns may dynamically affect the pressure gradient in the microchannel.

5:19PM E4.00004 Experimental droplet dynamics and interfacial rheology characterization , KENDRA ERK, NIST, JEFFREY MARTIN, Johnson & Johnson, JONATHAN SCHWALBE, Mitre, FREDERICK PHELAN JR., STEVEN HUDSON, NIST — Many properties of emulsions arise from interfacial rheology. Recently, a theory of droplet dynamics accounting for interfacial rheology, Marangoni forces and mass transport was developed. Here, we describe experimental observations of droplet dynamics in light of this theory. Using particle tracking velocimetry, we examine the dynamics of surfactant-stabilized droplets in the Poiseuille flow of a microfluidic device. Interfacial shear and dilatational properties are evaluated, and we distinguish viscous and elastic effects. We find that the shear viscosity of the interface populated by block copolymer surfactant is greater than with small molecule alcohol surfactant. Investigation of small droplets is of interest not only for their relevance to emulsion applications, but their small size has potential for improved force sensitivity and temporal resolution. Other droplet dynamic approaches will be discussed.

5:32PM E4.00005 Looking Under a Leidenfrost Drop , JUSTIN BURTON, AARON SHARPE, ROELAND VAN DER VEEN, ANDRES FRANCO, SIDNEY NAGEL, Univ. of Chicago — The Leidenfrost effect can be observed when small water drops move around effortlessly without sticking on a hot pan. The transition to a levitated state, where the drops rest on an insulating layer of vapor, occurs at the Leidenfrost temperature. Experiment [1] and theory [2] have examined the lifetime and maximum size of Leidenfrost drops. However, the liquid-vapor interface beneath the drop has not been fully charcterized. We report experiments using laser-light interference to measure the geometry of the liquid-vapor interface. By imaging the interference fringes produced between the bottom surface of the liquid and the hot substrate, we can measure the curvature of the vapor pocket beneath the drop as well as the azimuthal undulations along the neck that sits closest to the surface. From these measurements, we can extrapolate the shape of the bottom of the drop, which fluctuates in time with a period of a few milliseconds for millimeter-sized water drops. Our measurements of the azimuthal neck radius agree with predictions [2]: the difference between the drop and neck radii, $(R_d - R_n)\approx 0.53\lambda$ in the limit of large drops where λ is the capillary length of the fluid. For small drops we recover the result found in [1] that $R_n \propto R_d^2/\lambda$.

[1] A. Biance, C. Clanet, D. Quere, Phys. Fluids **15**, 1632 (2003).
[2] J. H. Snoeijer, P. Brunet, J. Eggers, Phys. Rev. E **79**, 036307 (2009).

5:45PM E4.00006 Experimental observations of droplet trajectories for investigation of water jet breakup and atomization , JOSHUA Z. SKOLNICK, LESTER K. SU, Johns Hopkins University — Experimental measurements of droplet trajectories permit investigation of the breakup and atomization processes in water jets. The jets issued at varying angles from the horizontal, with the resulting droplets being collected in a horizontal array of collection bins that provided good spatial resolution of droplet volume fluxes. The experiments encompassed varying jet diameters and exit velocities. Time-resolved imaging of the jet near-field provided information on the initial droplet size distributions, which was used in a model that generated predictions of droplet fluxes in the horizontal plane by Lagrangian tracking. The results of the experiments, interpreted with the assistance of the modeling results, allow direct assessment of the importance of secondary breakup phenomena at different run conditions. The results also reveal that droplet coalescence plays a significant role in determining the local droplet size distribution in the jet far field.

5:58PM E4.00007 Oscillatory flow measurements in the vicinity of pinned-contact capillary surfaces , JOSEPH OLLES, AMIR HIRSA, Rensselaer Polytechnic Institute — By coupling two pinned-contact droplets through a short tube, various devices have been demonstrated, including fast-focusing liquid lenses, pumps, and adhesion devices. The double droplet system (DDS) can be achieved on the millimeter scale with essentially spherical interfaces due to capillarity. A one-dimensional model based on the center of mass motion has been corroborated by experiments, but CFD simulations of the flow within the DDS (Ramalingam and Basaran, Phys. Fluids 2010) have yet to be experimentally validated. With PIV, we study the fluid flow in and around a DDS that is surrounded by an immiscible liquid. The flow near the corner and capillarity effects of the pinned-contact surfaces will be explored for different volumes of the DDS, oscillating frequencies, and driving amplitudes.

6:11PM E4.00008 The dynamics of impacting and coalescing Newtonian droplets[1] , JOSE RAFAEL CASTREJON-PITA, University of Cambridge, KRZYSZTOF J. KUBIAK, University of Leeds, ELEANOR S. BETTON, University of Cambridge, MARK C.T. WILSON, University of Leeds, ALFONSO A. CASTREJON-PITA, IAN M. HUTCHINGS, University of Cambridge, I4T PROJECT COLLABORATION — An experimental arrangement is described to study impact, coalescence and mixing between a sessile and an impacting droplet of glycerol/water striking a flat transparent substrate in air. We used high speed imaging to visualize and study the impact and coalescence of droplets from the side and from beneath. The impact parameters and liquid characteristics were chosen to match the typical dynamical conditions, based on Reynolds and Weber numbers, found in commercial drop-on-demand printing. The images were processed by particle image velocimetry and image processing algorithms to obtain velocity fields near the liquid-substrate interface, droplet geometries and contact line positions. The experimental results are compared with numerical simulations by the lattice Boltzmann method and good agreement is found. The experimental setup and its instrumentation are simple to reproduce and can be used in other practical applications.

[1]This project was supported by the EPSRC and industrial partners in the I4T and Glassjet projects.

6:24PM E4.00009 ABSTRACT WITHDRAWN –

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E5 Rayleigh-Benard Convection III: Turbulence 308

4:40PM E5.00001 Heat transport near the transition to the ultimate state of turbulent Rayleigh-Bénard convection[1] , GUENTER AHLERS, UC Santa Barbara, XIAOZHOU HE, MPIDS Göttingen, Germany, DENIS FUNFSCHILLING, LSGC CNRS Nancy, France, HOLGER NOBACH, EBERHARD BODENSCHATZ, MPIDS Göttingen, Germany — Measurements of the Nusselt number Nu for Rayleigh-Bénard convection (RBC) of a cylindrical sample over the Rayleigh-number range $2 \times 10^{12} \leq Ra \leq 2 \times 10^{15}$ and the Prandtl-number range $0.79 \leq Pr \leq 0.86$ are presented. The aspect ratio $\Gamma \equiv D/L$ was 0.50 ($D = 1.12$ m was the diameter and $L = 2.24$ m was the height). For $Ra \leq 2 \times 10^{13}$ the data yielded $Nu = Nu_0 Ra^{\gamma_{eff}}$ with $\gamma_{eff} = 0.312$, consistent with classical RBC.[2] For larger Ra γ_{eff} increased, reaching approximately $\gamma_{eff} = 0.38$ near $Ra = 10^{15}$, consistent with predictions[3,4] for an ultimate state with turbulent boundary layers.

[1]Supported by the Max Planck Society, the Volkswagenstiftung, and NSF Grant DMR07-02111.

[2]G. Ahlers, S. Grossmann, and D. Lohse, Rev. Mod. Phys. **81**, 503 (2009)

[3]R. H. Kraichnan, Phys. Fluids **5**, 1374 (1962)

[4]S. Grossmann and D. Lohse, Phys. Fluids **23**, 045108 (2011)

4:53PM E5.00002 Ultimate State of Two-Dimensional Rayleigh-Bénard Convection between Free-Slip Fixed-Temperature Boundaries[1] , JARED WHITEHEAD, CHARLES DOERING, University of Michigan — Rigorous upper limits on the vertical heat transport in two dimensional Rayleigh-Bénard convection between stress-free isothermal boundaries are derived from the Boussinesq approximation of the Navier-Stokes equations. The Nusselt number Nu is bounded in terms of the Rayleigh number Ra according to $Nu \leq 0.2891 Ra^{5/12}$ uniformly in the Prandtl number Pr. This Nusselt number scaling challenges some theoretical arguments regarding the asymptotic high Rayleigh number heat transport by turbulent convection.

[1]This research was supported in part by NSF DGE-0221611 and PHY-0855335. This work appears in Phys. Rev. Let. 106, 244501 (2011).

5:06PM E5.00003 The Reynolds number near the transition to the ultimate state of turbulent Rayleigh-Bénard convection[1] , XIAOZHOU HE, MPIDS, Goettingen, Germany, GUENTER AHLERS, University of California, Santa Barbara, DENIS FUNFSCHILLING, LSGC CNRS, Nancy, France, HOLGER NOBACH, EBERHARD BODENSCHATZ, MPIDS, Goettingen, Germany — Measurements of a Reynolds number Re for Rayleigh-Bénard convection (RBC) of a cylindrical sample over the Rayleigh-number range $2 \times 10^{12} \leq Ra \leq 2 \times 10^{15}$ and the Prandtl-number range $0.79 \leq Pr \leq 0.86$ are presented. The aspect ratio $\Gamma \equiv D/L$ was 0.50 ($D = 1.12$ m was the diameter and $L = 2.24$ m was the height). We used the elliptic approximation of He and Zhang[2,3] to determine an effective Re. For $Ra \geq 3 \times 10^{14}$ the data yielded $Re = Re_0 Ra^{\alpha_{eff}}$ with $\alpha_{eff} \simeq 0.50$. This result is consistent with predictions for the ultimate state, where the boundary layers are turbulent.[4,5]

[1]Supported by the Max Planck Society, the Volkswagenstiftung, and NSF Grant DMR07-02111.

[2]G.-W. He and J.-B. Zhang, Phys. Rev. **73**, 055303 (2006)

[3]X. He, G. He, and P. Tong, Phys. Rev. **81**, 065303 (2010)

[4]R. H. Kraichnan, Phys. Fluids **5**, 1374 (1962)

[5]S. Grossmann and D. Lohse, Phys. Fluids **23**, 045108 (2011)

5:19PM E5.00004 Ultimate Taylor-Couette turbulence , DETLEF LOHSE, SANDER HUISMAN, DENNIS VAN GILS, University of Twente, SIEGFRIED GROSSMANN, University of Marburg, CHAO SUN, University of Twente — With the Twente turbulent Taylor-Couette (T^3C) facility we can achieve $Re_i = 2 \cdot 10^6$ for the inner cylinder and $Re_o = \pm 10^6$ for the independently rotating outer cylinder. *Global* torque measurements have revealed an effective scaling $Nu_\omega \sim Ta^{0.38}$ for the dimensionless angular velocity flux [1]. We now provide *local* angular velocity flux measurements from high-speed particle image velocimetry (PIV). Though the flux shows huge fluctuations, its spatial and temporal average nicely agrees with the result from the global measurements and in addition reveals the structure of the turbulent flow. From the PIV images we can also derive the scaling of the radial velocity fluctuations, giving $Re \sim Ta^{1/2}$. Both Nu_ω and Re scaling relations exactly follow what had been predicted in ref. [2] for the ultimate regime.

[1] D. van Gils *et al.*, Phys. Rev. Lett. **106**, 024502 (2011).
[2] S. Grossmann and D. Lohse, Phys. Fluids **23**, 045108 (2011).

5:32PM E5.00005 Mixing of a stable linear density stratification in Taylor-Couette flow , R.L.F. OGLETHORPE, C.P. CAULFIELD, BPI & DAMTP, University of Cambridge, ANDREW W. WOODS, BPI, University of Cambridge — We consider mixing of an initially linear stable salt stratification in turbulent Taylor-Couette flow. The fluid is confined to a cylindrical annulus with a vertical axis. Mixing is caused by rotating the inner cylinder at a constant rate. The outer cylinder is fixed. Experimental measurements show that at high initial bulk Richardson number, defined as $Ri_0 = N^2/\Omega^2$, where N is the buoyancy frequency of the initial stratification and Ω is the rotation rate of the inner cylinder, an initially linear salt stratification develops a series of well mixed layers separated by sharp interfaces. The size of these layers appears to depend on Ri_0 and the gap width between the cylinders, Δ_R. With time, the layers at the top and bottom of the tank evolve in salinity. This leads to entrainment from and eventual mixing with the adjacent layers as the salinity contrast across these interfaces decreases. As a result of successive merger events, eventually the system becomes well mixed. The salinity of the inner layers appears to remain constant, so that salt is transported from the bottom layer to the top layer without changing the structure of the interior. The salt flux through an interface appears to depend only on the rotation rate Ω of the inner cylinder, consistent with our previous study for an initial two-layer salt stratification (Woods *et al.* (2010) *J Fluid Mech.* **663**, 347-357).

5:45PM E5.00006 Side wall effects in turbulent thermal convection , RICHARD J.A.M. STEVENS, DETLEF LOHSE, University of Twente, ROBERTO VERZICCO, University of Rome — Research in thermally driven convective turbulence has always been trying to achieve the highest Rayleigh number (Ra) as various natural heat transfer phenomenon involve very high Ra. Since seemingly equivalent experiments with the same Ra and Prandtl Pr numbers and cell aspect–ratio (Γ) have not given the same heat transfers in the high–end of Ra, it has been conjectured that neglected details, like the finite conductivity of horizontal plates and sidewall, could play a role. It has indeed been shown that corrections for the heat leakage through the sidewall and for the finite thermal conductivity of the plates must be considered, although none of them could fully explain the differences among the experiments. Ahlers et al. recent experiments suggest that a factor that could trigger different heat transfers regimes is the temperature *outside* the cell. Indeed, in real experiments the sidewall is isothermal, but in most models it is assumed to be adiabatic. In our numerical simulations we now allow for an isothermal sidewall and assess the behavior of the system in presence of a conductive sidewall. We discuss the differences between an ideal set–up with perfectly adiabatic side walls and systems with an isothermal sidewall or a sidewall with finite thickness and heat conductivity.

5:58PM E5.00007 Boussinesq and non-Boussinesq plumes - universal solutions , TON S. VAN DEN BREMER, GARY R. HUNT, Department of Civil and Environmental Engineering, Imperial College London — Closed-form solutions describing the behaviour of rising buoyant axisymmetric turbulent plumes, emitted vertically from area sources in unconfined quiescent environments of uniform density, are proposed in a form that is universally applicable to Boussinesq and non-Boussinesq plumes. This work [1] thereby generalizes and consolidates results obtained separately for Boussinesq and non-Boussinesq plumes both based on the seminal plume model of Morton et al (1956) [2]. The universality we identify crucially relies on expressing the plume equations in terms of the vertical variation of the plume Richardson number and the dimensionless radius, the introduction of an effective entrainment radius, and the classical entrainment model for which the horizontal entrainment velocity is proportional to the local vertical velocity; this velocity being modified in the non-Boussinesq case by a factor of $\sqrt{\rho/\rho_a}$, where ρ and ρ_a are the densities of the plume and the ambient fluids, respectively. Using the solutions developed, three characteristic heights are identified.

[1] Van den Bremer, T.S. & Hunt, G.R., 2010. Universal solutions for Boussinesq and non-Boussinesq plumes. *J. Fluid Mech.* **644**, pp. 165 - 192.
[2] Morton, B.R., Taylor, G.I. & Turner, J.S., 1956. Turbulent gravitational convection from maintained and instantaneous sources. *Proc. Roy. Soc. Lond. A* **234**, pp. 1 - 23.

6:11PM E5.00008 Quantifying near-wall coherent structures in turbulent convection , G.S. GUNASEGARANE, BABURAJ A PUTHENVEETTIL, IIT Madras, YOGESH K AGRAWAL, NIT Durgapur, India, DANIEL SCHMELING, JOHANNES BOSBACH, Institute of Aerodynamics and Flow Technology, German Aerospace Center (DLR), JAYWANT H. ARAKERI, IISc Bangalore, IIT MADRAS- DLR - IISC COLLABORATION — We present planforms of line plumes formed on horizontal surfaces in turbulent convection, along with the length of near- wall line plumes measured from these planforms, in a six decade range of Rayleigh numbers ($10^5 < Ra < 10^{11}$) and at three Prandtl numbers ($Pr = 0.7, 6, 602$). Using geometric constraints on the relations for the mean plume spacings, we obtain expressions for the total length of these near-wall plumes in turbulent convection. The plume length per unit area (L_p/A), made dimensionless by the near-wall length scale in turbulent convection (Z_w) remains a constant for a given fluid. The Nusselt number is shown to be directly proportional to L_pH/A for a given fluid layer of height H. Increase in Pr has a weak influence in decreasing L_p/A. These expressions match the measurements, thereby showing that the assumption of laminar natural convection boundary layers in turbulent convection is consistent with the observed total length of line plumes. We then show that similar relationships are obtained based on the assumption that the line plumes are the outcome of the instability of laminar natural convection boundary layers on the horizontal surfaces.

6:24PM E5.00009 Plume Statistics in Turbulent Rayleigh-Bénard Convection[1] , JAMES HOGG, University of California, Santa Barbara, JOHANNES BOSBACH, DLR, Goettingen Germany, STEPHAN WEISS, GUENTER AHLERS, University of California, Santa Barbara — We report spatial statistics of thermal plumes based on an analysis of shadowgraph images for a cylindrical Rayleigh-Bénard sample of height $L = 9.5$ mm and aspect ratio $\Gamma = 10.6$. The fluids consisted of pure gases with Prandtl- numbers $Pr \approx 0.7$ and gas mixtures with $0.17 \leq Pr < 0.7$. The Rayleigh-number range was $3 \times 10^5 \leq Ra \leq 5 \times 10^6$. We found the plume-length probability-density functions to be well fit by lognormal distributions with fit parameters which revealed a sharp transition from patterns to turbulence. In the turbulent range the mean spacing between plumes was found to scale as $Ra^{-0.33}$, suggesting that the Kolmogorov length η_K is the relevant length scale, rather than the thermal boundary-layer thickness.

[1] Supported by NSF Grant DMR07-02111.

6:37PM E5.00010 Experimental Measurement of the Local Energy Dissipation Rate and its Balance with the Local Heat Flux in Turbulent Thermal Convection[1] , SHI-DI HUANG, RUI NI, KE-QING XIA, Department of Physics, The Chinese University of Hong Kong — The local energy dissipation rate $\epsilon_{u,c}$ in Rayleigh- Bénard convection cell was measured experimentally using the particle tracking velocimetry method, with varying Rayleigh number Ra, Prandtl number Pr, and system size H. It is found that $\epsilon_{u,c}/(\kappa^3 H^{-4}) = 1.05 \times 10^{-4} Ra^{1.55\pm0.02} Pr^{1.15\pm0.38}$. The Ra- and H-dependency of the measured result are found to be consistent with the assumption made for the bulk energy dissipation rate $\epsilon_{u,bulk}$ in the Grossmann-Lohse model, while the Pr-dependency is not. A remarkable finding of the study is that $\epsilon_{u,c}$ balances the directly measured local Nusselt number Nu$_c$ in the cell center, not only scaling-wise but also in magnitude.

[1] This work was supported by RGC of Hong Kong SAR (No. CUHK403807 and 404409).

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E6 Low Reynolds Number Swimming I 309

4:40PM E6.00001 Time-dependent flow fields around the spherical colonial alga *Volvox carteri*

, DOUGLAS BRUMLEY, MARCO POLIN, CONSTANT MOREZ, RAYMOND GOLDSTEIN, TIMOTHY PEDLEY, DAMTP, University of Cambridge — *Volvox carteri* is a spherical colonial alga, consisting of thousands of biflagellate cells. The somatic cells embedded on the surface of the colony beat their flagella approximately towards the south pole, producing a net fluid motion. Using high-speed imaging and particle image velocimetry (PIV) we have been able to accurately analyse the time-dependent flow fields around such colonies. The somatic cells on the colony surface may beat their flagella in a perfectly synchronised fashion, or may exhibit behaviour in which the coordination wanders periodically between forward and backward propagating metachronal waves. We analyse the dependence of this synchronisation on fundamental parameters in the system such as colony radius, characterise the speed and wavelength of metachronal waves propagating on the surface, and investigate the extent to which hydrodynamic interactions are responsible for the exhibited behaviour. The time-averaged flow fields agree with previous experiments involving freely swimming colonies (Phys. Rev. Lett. 105:168101, 2010) and Blake's squirmer model (J. Fluid Mech. 46, 199-208, 1971b).

4:53PM E6.00002 Particle Image Velocimetry Around Swimming Paramecia

, MATTHEW GIARRA, SAIKAT JANA, SUNGHWAN JUNG, PAVLOS VLACHOS, Virginia Tech — Microorganisms like paramecia propel themselves by synchronously beating thousands of cilia that cover their bodies. Using micro-particle image velocimetry (μPIV), we quantitatively measured velocity fields created by the movement of *Paramecium multimicronucleatum* through a thin ($\sim$100 μm) film of water. These velocity fields exhibited different features during different swimming maneuvers, which we qualitatively categorized as straight forward, turning, or backward motion. We present the velocity fields measured around organisms during each type of motion, as well as calculated path lines and fields of vorticity. For paramecia swimming along a straight path, we observed dipole-like flow structures that are characteristic of a prolate-spheroid translating axially in a quiescent fluid. Turning and backward-swimming organisms showed qualitatively different patterns of vortices around their bodies. Finally, we offer hypotheses about the roles of these different flow patterns in the organism's ability to maneuver.

5:06PM E6.00003 High Speed Tomographic PIV Measurements of Copepod Sensory Cues

, DONALD R. WEBSTER, DAVID MURPHY, School of Civil and Environmental Engineering, Georgia Institute of Technology, JEANNETTE YEN, School of Biology, Georgia Institute of Technology — A steady siphon flow is commonly used to mimic the aquatic suction feeding of piscine predators in studies of zooplankton sensory ecology. The sensory and escape behavior of copepods, with their long, highly enervated, setae-bearing antennules, has been investigated using this prescribed flow field, modeled analytically as a point sink. The position of the animal when it escapes provides a threshold for the species-specific strain rate value (as low as 0.4 s^{-1}) that evokes this ecologically-important behavior. Understanding the actual mechanics of copepod sensing, however, requires more than just correlation analysis based on position in the strain rate field. Knowledge of the setae-bending flow field along the length of the antennules during the time leading up to the escape jump is needed to fully understand the sensory mechanism. Measurements of this type have not been practical using traditional planar PIV techniques. We present time-resolved tomographic PIV measurements of the flow around the sensory appendages of *Acartia* spp. copepods responding to siphon flow and provide insight into copepod behavior in response to fluid mechanical stimuli.

5:19PM E6.00004 A rigorous proof of the scallop theorem and finite mass effects of a microswimmer

, KENTA ISHIMOTO, MICHIO YAMADA, Research Institute for Mathematical Sciences, Kyoto University — We reconsider fluid dynamics of a self-propulsive swimmer in Stokes flow. With an exact definition of deformation of a swimmer, a proof is given to the scallop theorem including the body rotation of the swimmer. Introducing a virtual swimmer, which has the same shape as the real swimmer but has no ambient fluid, we give an exact definition of the surface deformation of the generally translating and rotating swimmer, and then prove the scallop theorem rigorously for the massless swimmer. We also discuss the breakdown of the scallop theorem due to a finite mass (finite Stokes number) of the swimmer by using a perturbation expansion method and it is found that the breakdown generally occurs at the first order of Stokes number. In addition, employing the Purcell's "scallop" model, we show that the theorem holds up to a higher order of the Stokes number, if the swimmer's stroke has some symmetry.

[1] K. Ishimoto and M. Yamada, "A rigorous proof of the scallop theorem and a finite mass effect of a microswimmer," arXiv:1107.5938v1 [physics.flu-dyn].

5:32PM E6.00005 Performance of flexible low-Re swimmers in Newtonian and viscoelastic liquids

, J. ESPINOSA, R. ZENIT, Universidad Nacional Autonoma de Mexico, E. LAUGA, University of California - San Diego — We show experimental results of "flexible tail" swimmers in elastic fluids. A magnetic microswimmer powered by a frequency-controlled homogeneous magnetic field was built. Experiments were performed in a reference viscous Newtonian fluid and a glucose-based Boger fluid of the same shear viscosity. High definition video of the swimmer traveling along a channel was taken to measure its average swimming speed. We found that locomotion is enhanced in elastic fluids for most conditions. To further investigate the swimming performance, the flow field around the swimmer was visualized with a PIV (Particle Image Velocimetry) technique. The differences between Newtonian and Boger fluid will be presented and discussed.

5:45PM E6.00006 Locomotion of helical swimmers in Newtonian and complex fluids

, F.A. GODINEZ, R. ZENIT, Universidad Nacional Autonoma de Mexico, E. LAUGA, University of California - San Diego — A biomimetic swimming device was built using a magnetic head and a rigid helical tail to investigate experimentally the effects of viscoelasticity on creeping flow locomotion. The millimeter-size swimmer is controlled and actuated wirelessly using a rotating magnetic field. Two fluids with similar viscosities were used for experiments, one Newtonian and one viscoelastic Boger fluid. Preliminary results from both fluids show a linear relationship between the frequency of the applied field and the translational velocity. The objective of this investigation is to determine if the viscoelasticity of the fluids enhances or not the swimming performance for this device.

5:58PM E6.00007 On growth and flow: bacterial biofilms in porous media

, WILLIAM DURHAM, ALBERTO LEOMBRUNI, OLIVIER TRANZER, ROMAN STOCKER, MIT — Bacterial biofilms often occur in porous media, where they play pivotal roles in medicine, industry and the environment. Though flow is ubiquitous in porous media, its effects on biofilm growth have been largely ignored. Using patterned microfluidic devices that simulate unconsolidated soil, we find that the structure of *Escherichia coli* biofilms undergoes a self-organization mediated by the interaction of growth and flow. Intriguingly, we find that biofilm productivity peaks at intermediate flow rates, when the biofilm is irrigated by a minimum number of preferential flow channels. At larger and smaller flow rates, fluid flows more uniformly through the matrix, but productivity drops due to removal by shear and reduced nutrient transport, respectively. These dynamics are correctly predicted by a simple network model. The observed tradeoff between growth and flow may have important consequences on biofilm-mediated processes such as biochemical cycling, antibiotic resistance and water filtration.

6:11PM E6.00008 Maneuverability at low Reynolds number[1]

, LISA BURTON, Massachusetts Institute of Technology, ROSS HATTON, Carnegie Mellon University, HOWIE CHOSET, Carnegie Mellon Univeristy, A.E. HOSOI, Massachusetts Institute of Technology — Speed and efficiency are common and often adequate measures to compare swimming systems. However, these metrics do not take into account how well a system can reconfigure, change direction, or move in a confined environment. Inspired by manipulability, a concept used to analyze robotic arms, we propose new metrics to consistently quantify the maneuverability of a system. We discuss a general definition of maneuverability and apply it to low Reynolds number swimming. Additionally, we identify the library of motions to maximize an artificial swimmer's maneuverability and explore the effect of morphology and kinematics on maneuverability.

[1] Battelle Memorial Institute

6:24PM E6.00009 Microswimmers in Complex Environments with Heterogeneous Microstructure , YUNKYONG HYON, HENRY FU, University of Nevada, Reno — We will discuss the swimming of microorganisms in complex and heterogeneous environments. Microswimmers in biological complex fluids, for instance, bacteria and sperm, are often greatly influenced by heterogeneous medium microstructure with length scales comparable to themselves. We characterize the interaction between the microswimmer and the medium microstructure using the model Golestanian three-sphere swimmer, treating the hydrodynamic interaction with the microstructure through the Oseen tensor. In this investigation, the microstructure of the heterogeneous environment is modeled by fixed spheres representing obstacles, or chains consisting of spheres connected with elastic springs. We find that the swimming speed of the swimmer depends on the force and deformation exerted on micro-structure. Furthermore, we find that while short freely suspended chains and short chains anchored at their ends interact with swimmer quite differently, long enough chains interact similarly, that is, a long mobile chain acts like a anchored chain. We discuss the implications for swimmer interactions with polymer solutions and compliant networks.

6:37PM E6.00010 Swimming speed of an oscillating sheet in Newtonian and viscoelastic fluids , MOUMITA DASGUPTA, Clark University, MICHAEL BERHANU, Laboratoire MSC Université Paris Diderot, ARSHAD KUDROLLI, Clark University, BIN LIU, KENNETH BREUER, THOMAS POWERS, Brown University — We discuss a mechanical experimental model of a flexible sheet swimming with a prescribed wave pattern through a fluid. We are motivated by a need for a fundamental understanding of microorganism locomotion through non-Newtonian fluids. To simplify the problem, we suspend a tall flexible cylindrical sheet concentric within a cylindrical tank filled with the fluid. Torque free boundary conditions are imposed by supporting the flexible sheet and the tank with friction-free ball-bearings. A traveling wave is imposed on the sheet with a pair of rollers in the azimuthal direction. We first show that the swimming speed is linear with respect to the phase velocity of the traveling wave for a viscous Newtonian fluid. Then we show that the system is essentially two dimensional as the results do not depend on the height of fluid in the tank. We measure swimming speed in Polyox-water mixtures and Sodium CMC solutions as a function of wave speed. We again demonstrate linear response in the swimming speeds, which also decrease in these viscoelastic fluids relative to the Newtonian case as wave speed increases. Decrease in swimming speed is observed with increase in viscoelasticity of the fluids. We then discuss the dependence of swimming speed on the Deborah number of the fluids.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E7 Turbulent Boundary Layers III: Roughness 310

4:40PM E7.00001 A Study of the Mean Force Structure of Rough-Wall Turbulent Boundary Layers[1] , FARAZ MEHDI, JOSEPH KLEWICKI, University of New Hampshire — Analysis of existing data by Mehdi, Klewicki & White [Physica D 239(2010)] provides evidence that the traditional classifications do not fully account for the combined effects of roughness and Reynolds number. We continue to explore this further, and in the present talk report on experiments that used 24-grit sandpaper and pea gravel for roughness over an 8m fetch. Two-component LDV measurements are used to acquire well-resolved mean velocity and Reynolds stress profiles over a modest range of Reynolds numbers. These data are used to estimate the terms in the appropriate mean statement of dynamics, which directly reveals the operative time-averaged balance of forces. The present results further reinforce the previous observation that the mean viscous force retains dominant order above (and often well-above) the roughness elements. Force balance data are shown to be usefully organized relative to the length scale that defines the region from the wall to where the leading order mean dynamics are described by a balance between mean advection and the mean effect of turbulent inertia. In the smooth-wall flow, this length scale is only a function of Reynolds number. In rough-wall flows, the data indicate it to be a function of roughness and Reynolds number.

[1]The support of the ONR (N000140810836, grant monitor Ronald Joslin) is gratefully acknowledged.

4:53PM E7.00002 Scale-separation phenomena in rough-wall turbulent boundary layers[1] , RACHEL EBNER, CALEB MORRILL-WINTER, University of New Hampshire, JOSEPH KLEWICKI, University of New Hampshire, University of Melbourne — Analysis of data from smooth-wall flows indicates that scale- separation and scale-selection phenomena occur between the velocity and vorticity fields as a function of distance from the wall and Reynolds number. These phenomena are significant since correlations between specific velocity and vorticity components underlie the momentum and energy transport mechanisms operative in boundary layer turbulence. The current effort explores these phenomena in rough-wall flows by examining the behaviors of velocity and vorticity component spectra and co-spectra as derived from multi-sensor hot-wire measurements. The data were acquired at multiple Reynolds numbers and for two different sizes of homogeneously distributed roughness. A four-wire probe configuration allowed for well-resolved measurements of the streamwise and wall-normal velocity components, as well as the spanwise component of vorticity. Pre-multiplied power spectra and the associated co-spectra are interpreted relative to the wall- normal variations of the Reynolds stress and kinetic energy gradient. These results are compared to the smooth-wall analysis, and the recent rough-wall findings of Hong et al. (*J. Fluid Mech.* **667**, 2011) indicating that roughness effectively sets the scale of the vortical motions throughout the boundary layer.

[1]The support of the ONR (N000140810836, grant monitor Ronald Joslin) is gratefully acknowledged.

5:06PM E7.00003 Spatial Characteristics of Large-Scale Motions in Smooth and Rough Turbulent Boundary Layers , J.M. BARROS, K.T. CHRISTENSEN, MechSE Dept., Univ. of Illinois — Wide field of view stereo PIV measurements were conducted in the wall-normal–spanwise plane of smooth and rough zero-pressure- gradient turbulent boundary layers. The roughness under consideration was replicated from a turbine blade damaged by deposition of foreign materials and contained a broad range of scales arranged in an irregular manner. Inspection of smooth- and rough-wall instantaneous velocity fields in this cross plane uncovered the spatial signatures of low- and high-momentum regions, the former of which have been previously linked to hairpin vortex packets in the outer layer of wall turbulence. Proper orthogonal decomposition was employed to study the spatial characteristics of the large-scale motions in smooth- wall flow and revealed a significant spanwise coherence in the form of alternating low- and high-momentum regions well beyond that reflected in two-point velocity correlations. In contrast, the rough-wall results revealed weaker coherence of these large- scale motions.

5:19PM E7.00004 The Impact of Favorable-Pressure-Gradient Conditions on Large-Scale Motions in Smooth- and Rough-Wall Turbulence , D. MIN, K.T. CHRISTENSEN, MechSE Dept., Univ. of Illinois — The combined impact of irregular surface roughness and moderate favorable-pressure-gradient (FPG) conditions ($K \approx 4.0 - 4.4(10^{-7})$) on large-scale motions in a turbulent boundary layer was assessed using stereo PIV measurements in the wall- normal–spanwise plane. The roughness under consideration was replicated from a turbine blade damaged by deposition of foreign materials and contains a broad range of topographical scales. These measurements were compared to measurements of smooth-wall flow under identical FPG conditions to reveal the combined impact of roughness and FPG conditions on the larger-scale motions. Instantaneous smooth- and rough- wall velocity fields embodied spanwise-alternating patterns of low- and high-momentum regions (LMRs and HMRs). While these motions in the smooth-wall case were found to penetrate less deep into the boundary layer compared to similar motions in zero- pressure-gradient flow, they often extended much further away from the wall in the presence of roughness. Two-point correlations of low-pass- filtered velocity fields embodying only the larger-scale motions of smooth-wall flow revealed a remarkable spanwise- alternating nature of low- and high-momentum regions well beyond that reflected in correlations of unfiltered velocity. Roughness was found to reduce this spanwise coherence.

5:32PM E7.00005 Large eddy simulation of accelerating turbulent boundary layers over rough surfaces , JUNLIN YUAN, UGO PIOMELLI, Queen's University — Rough-wall turbulent boundary layers subject to favourable pressure gradients (FPGs) and freestream acceleration are found in many engineering applications. On a smooth wall with a strong FPG the flow may revert to a quasi-laminar state. Roughness, on the other hand, enhances turbulent fluctuations near the wall. We investigate strongly accelerating turbulent boundary layers over rough surfaces by large eddy simulation. We use an immersed-boundary method to model random sandgrain-type roughness. Flow statistics show that the extent of relaminarization depends on the degree of acceleration, on the roughness height, and on the flow Reynolds number. The stabilizing effect of the FPG (laminar-like mean velocity, decreased Reynolds stresses, increased anisotropy) is only found outside the roughness sublayer: close to the wall, roughness results in an increase of all components of the Reynolds stress and decreases Reynolds-stress anisotropy. An increase in the Reynolds number intensifies the roughness effect. Small eddies are generated in the wake of the roughness elements; they tend to disrupt the streak stabilization observed in FPG boundary layers, and increase bursting. This study confirms previous findings which indicate that roughness competes with the FPG stabilizing effect near the wall, and might deactivate acceleration effect in the outer layer.

5:45PM E7.00006 HPIV based volumetric 3D flow description in the roughness sublayer of a turbulent channel flow[1] , SIDDHARTH TALAPATRA, JOSEPH KATZ, Department of Mechanical Engineering, The Johns Hopkins University — Microscopic HPIV is utilized to resolve the 3D flow in the roughness sublayer of a boundary layer over a rough wall at Re_τ=3400, consisting of pyramidal elements with height of k=0.45mm and 3.3mm wavelength. Typically, ~7000 particles are tracked in a 3.2×2.1×1.8mm^3 volume, whose wall-normal extent is -0.2< y/k <4.67, y=0 being the roughness peak. These measurements are facilitated by matching the refractive index of the fluid with that of the acrylic rough wall. Results show that the sublayer is flooded by complex coherent structures scaled between 1-2k. They are mostly aligned with roughness grooves, but some wrap around the pyramids, and stretch to a streamwise orientation by a relatively fast channeling flow that develops between the pyramid ridgelines. Occasionally, structures eject away from the roughness sublayer at a steep angle to the mean flow. Using the 300 realizations processed so far, the spatial variations in mean velocity and Reynolds stresses are compared to 2D PIV results, and trends generally (but not always) agree. In particular, there is a rapid increase in all Reynolds stress components close y=0. Conditional sampling is used to extract statistically significant structures.

[1]Sponsored by ONR (grant No. 000140-91-10-0-7).

5:58PM E7.00007 Coherent structures and associated sub-grid scale energy transfer in a rough-wall turbulent channel flow[1] , JIARONG HONG, JOSEPH KATZ, CHARLES MENEVEAU, Johns Hopkins University, MICHAEL SCHULTZ, United States Naval Academy — Our earlier study has revealed spectral bumps and enhanced dissipation of turbulent kinetic energy (TKE) by roughness scale eddies in the outer parts of a rough wall channel flows at Re_τ=3520-5360. Here, conditional averaging of PIV data, based on high subgrid-scale (SGS) energy flux, shows an inclined large structure with negative spanwise vorticity in streamwise-wall-normal (xy) planes, and two pairs of counter-rotating vortices in streamwise-spanwise (xz) planes, both near the wall and in the outer-layer. In instantaneous snapshots, this structure consists of periodic bundles of roughness-scale eddies spaced by a roughness wavelength, resulting from lifting and re-orientation of near-wall, quasi-streamwise vortices. Near the wall, the peak of SGS energy flux occur at the intersection of the quasi-streamwise eddies, while in the outer-layer, elevated flux occurs within the large structure owing to interaction among eddies in a bundle. At the conditioning point, the SGS kinetic energy peaks, but the resolved TKE has a minimum. The location of the resolved TKE maximum varies with elevation due to the increasing impact of ejections with distance from the wall.

[1]Sponsored by ONR.

6:11PM E7.00008 POD Analysis of a ZPG Turbulent Boundary Layer with and without Surface Roughness , JENSEN NEWMAN, PAT RUSSO, LUCIANNO CASTILLO, Rennselaer Polytechnic Institute, GUILLERMO ARAYA, Swansea University, DON DREW, Rennselaer Polytechnic Institute — Effects of surface roughness on coherent structures of a zero pressure gradient turbulent boundary layer are investigated via the POD method. In this study we seek to gain better understanding of how the small scales of turbulence are able to influence the large scales; particularly in rough surface flows. Comparison with coherent structures in a smooth case by Baltzer et al (2010) will be shown to assess the role of surface roughness in a transitional rough regime. The POD analysis is performed using the method of snapshots on two turbulent boundary layer DNS databases created using the dynamic multi scale recycling plane method developed by Araya et al. (2011) at Reynolds numbers of 2000-2300. Snapshots are separated by a time of $150\theta_0/U_\infty$ Resulting POD modes are qualitatively compared to assess the effects of roughness on the coherent structures. Baltzer, R.J.; Adrian, J.; Wu, X., "Turbulent Boundary Layer Structure Identification via POD," in *Stanford Center for Turbulence Research Summer Program*, 2010. G. Araya, *et al.*, "A dynamic multi-scale approach for turbulent inflow boundary conditions in spatially developing flows," *Journal of Fluid Mechanics,* vol. 670, pp. 581-605, 2011.

6:24PM E7.00009 Scaling of transitionally-rough flow structures in DNSes of channels with riblets up to $Re_\tau \approx 550$, GARCÍA-MAYORAL RICARDO, JAVIER JIMÉNEZ, U. Politécnica Madrid — The linear, drag-reducing effect of vanishingly small riblets breaks down once their size is in the transitionally-rough regime. We have previously reported that this breakdown is caused by the additional Reynolds stresses produced by the appearance of elongated spanwise rollers just above the riblet surface. These rollers are related with the Kelvin–Helmholtz instability of free shear layers, and to similar structures appearing over other rough and porous surfaces. However, because of the limited $Re_\tau \approx 180$ in our previous DNSes, it could not be determined whether those structures scaled in inner or outer units. Furthermore, it is questionable if results in the transitionally-rough regime at $Re_\tau \approx 180$ can be extrapolated to configurations of practical interest. At such small Reynolds numbers, roughness of transitional size can perturb a large portion of the boundary layer, which is not the case in most industrial and atmospheric applications. To clarify these issues we have conducted a set of DNSes at $Re_\tau \approx 550$. Our results indicate that the spanwise rollers scale in wall units, and support the validity of the extrapolation to configurations of practical interest.

6:37PM E7.00010 Optimization of drag-reducing surface geometries in turbulent channel flow[1] , ASGHAR YARAHMADI, MEREDITH METZGER, University of Utah — Turbulent channel flow over surfaces comprised of spanwise-wavy arrays of longitudinally-oriented riblets was simulated computationally to determine the optimal riblet configuration yielding maximum drag reduction. Surface riblet patterns were characterized by geometric parameters including riblet height, spacing, and cross-sectional shape (i.e., side-wall curvature/slope) as well as the spanwise amplitude and wavelength of the sinusoidal undulation. This parameter space allowed investigation of the following types of riblet surfaces: rectangular, triangular, trapezoidal, notched top, scalloped semicircular, and U-shaped. The surface configuration affording the highest cost savings in terms of skin friction drag reduction was obtained using a multi-objective evolutionary optimization method driven by numerical simulations that were performed using LES with a localized dynamic SGS model, RANS (RSTM and two equations methods) and Detached Eddy Simulation model. The computational efficiency and accuracy of the turbulence models are also discussed.

[1]Support by Utah's Center for High Performance Computing is gratefully acknowledged.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E11 Turbulence: Mixing I 314

4:40PM E11.00001 Direct numerical simulation of scalar transport in turbulent flows over progressive water waves , DI YANG, LIAN SHEN, Johns Hopkins University — The transport of passive scalars in turbulent flows over progressive water waves is studied using direct numerical simulation. A hybrid pseudo-spectral and finite-difference scheme on a wave-surface-fitted grid is used to resolve the flow and scalar fields near the wave surface. Various Schmidt numbers, wave phase speeds, and wave slopes are considered, and their effects on the instantaneous and statistics of scalar concentration are investigated. The scalar field shows apparent wave-phase dependence induced by the wave surface curvature and motions. For slow waves, the scalar field is mainly affected by the wave surface geometry with similarities with the case of a stationary wavy boundary. Differently, when the wave phase speed is large, the wave motion has strong influence on the scalar field. Budget for the intensity and flux of scalar concentration fluctuations are also studied. The analysis provides useful insight into the modeling of scalar transport over progressive waves.

4:53PM E11.00002 Turbulent mixing in a river confluence controls the fate of contaminated sediments , CRISTIAN ESCAURIAZA, CHRISTIAN GONZALEZ, PAULA GUERRA, PABLO PASTEN, GONZALO PIZARRO, Hydraulic and Environmental Engineering Department, Pontificia Universidad Catolica de Chile — A river confluence in a 40-degree angle and a high momentum ratio (M=12.8) mixes water with different chemical compositions favoring the formation of arsenic-rich particles. The dynamics of turbulent coherent structures at the mixing interface between both streams controls the transport and fate of these particles. Motivated by these field measurements, we integrate an experimental investigation in the laboratory with numerical computations based on the detached-eddy simulation (DES) approach. Streamwise helical vortices and shear layers have a significant effect on the size and deposition of the particles and their interaction with bed sediment. Through this study we provide new insights on the dynamics of coherent structures in confluences and the influence of turbulent mixing on chemical processes and contaminant fate in aquatic environments.

5:06PM E11.00003 Cumulative compressibility effects on population dynamics in turbulent flows , PRASAD PERLEKAR, Department of Applied Physics, Eindhoven University of Technology, DAVID R. NELSON, Department of Physics, Harvard University, Cambridge, USA, ROBERTO BENZI, Department of Physics, University of Tor Vergata, Roma, Italy, FEDERICO TOSCHI, Department of Applied Physics, Eindhoven University of Technology — Bacteria and plankton populations living in oceans and lakes reproduce and die under the influence of turbulent currents. Turbulent transport can interact in a complex way with the dynamics of populations because the typical reproduction time of microorganism is well within the inertial range of turbulence time scales. We quantitatively investigate the effect of flow compressibility on the dynamics of populations. While a small compressibility can be induced by several physical mechanisms, like density mismatch or the finite size of microorganisms with respect to the fluid turbulence, its effect on the the carrying capacity (average population concentration) of the ecosystem can be dramatic. We show that a tiny compressibility can produce a finite effect on the carrying capacity, this is due to an integrated effect made possible by the long replication times of the organisms with respect to turbulence time scales. We also present a full statistical quantification of the fluctuations of the population concentration that leads to a data collapse over a broad range in parameter space.

5:19PM E11.00004 Effect of LES models on the entrainment of a passive scalar in a turbulent planar jet , DIOGO CHAMBEL LOPES, CARLOS DA SILVA, RICARDO REIS, IST - Technical University of Lisbon, VENKAT RAMAN, UT Austin — Direct and large-eddy simulations (DNS/LES) of turbulent planar jets are used to study the role of subgrid-scale models in the integral characteristics of the passive scalar mixing in a jet. Specifically the effect of subgrid-scale models in the jet spreading rate and centreline passive scalar decay rates are assessed and compared. The modelling of the subgrid-scale fluxes is particularly challenging in the turbulent/nonturbulent (T/NT) region that divides the two regions in the jet flow: the outer region where the flow is irrotational and the inner region where the flow is turbulent. 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. 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.

5:32PM E11.00005 An advection-diffusion model for the dispersion in quasi two-dimensional steady turbulent jets , JULIEN R. LANDEL, C.P. CAULFIELD, DAMTP & BPI, University of Cambridge, ANDREW W. WOODS, BPI, University of Cambridge — The study of turbulent jets in relatively enclosed geometries is relevant to many chemical engineering processes. Predicting the concentration of chemical reactants in time and space requires a good understanding of the jet dynamics. We have considered experimentally and theoretically the behaviour of liquid jets in a quasi-Hele-Shaw cell, where the jets are constrained in a narrow gap whose width is two orders of magnitude smaller than the length-scales of the other two flow dimensions. In this configuration, the dynamics shown by the jets is very rich. Detailed examinations of instantaneous structures of the flow reveal a high-speed sinuous core at the centre of the jet and large vortical structures on each side, which we analyse quantitatively using a variety of techniques (particle image velocimetry and dye experiments). These structures have a large impact on the mixing and dispersion properties of the jet. We propose a one-dimensional advection-diffusion model to account for the vertical dispersion in the jet. The diffusion coefficient assumed in the model is based on our understanding of the large-scale structures of these jets. The model is solved analytically using a similarity form in the case of a finite-volume release of tracers in the jet. The theoretical predictions and the experimental measurements show very good agreement.

5:45PM E11.00006 Effect of background turbulence on an axisymmetric turbulent jet , LAURENT MYDLARSKI, Department of Mechanical Engineering, McGill University, BABAK KHORSANDI, SUSAN GASKIN, Department of Civil Engineering and Applied Mechanics, McGill University — An experimental investigation into the effect of different levels of background turbulence on the dynamics of a momentum-driven, axisymmetric, turbulent jet was conducted. An approximately homogenous isotropic turbulent background with zero mean flow was generated using a random jet array. The velocities were measured by acoustic Doppler velocimetry and flying hot-film anemometry. The results show that the mean axial velocities decay faster in the presence of background turbulence, while the mean radial velocities increase, especially close to the edges of the jet. The RMS velocity of the jet issuing into the turbulent background also increased compared to that of a jet emitted into a quiescent background. In addition, the width of the jet increased in the presence of the background turbulence. The mass flow rate of the jet decreased in the presence of the background turbulence, from which it can be inferred that the entrainment into the jet is reduced. It is hypothesized that, in the presence of background turbulence, large-scale engulfment (and not small-scale nibbling) is expected to be the main entrainment mechanism.

5:58PM E11.00007 Uncertainty of RANS mixing model prediction for an underexpanded jet in a supersonic cross flow[1] , CATHERINE GORLE, Center for Turbulence Research, Stanford University, GIANLUCA IACCARINO, Stanford University — The complex phenomena involved in scramjet propulsion can be investigated using RANS simulations, but the simulations do not fully represent all the physics involved. The mixing of fuel and air inside the supersonic combustion chamber is one of the critical processes that requires modeling. The goal of the present work is the development of an uncertainty model for the RANS modeling of mixing that can be used to characterize the safety and operability limits of scramjet engines. To this end, RANS simulations of an underexpanded jet in a supersonic cross flow are considered. Experimental data and an LES database are available from literature. The uncertainties in the RANS simulations are related to the models for the Reynolds stresses and the mixture fraction fluxes. The sensitivity of the model outcome to all sources of uncertainty in these models is characterized and quantified by introducing perturbations in the Reynolds stresses and the mixture fraction fluxes. Through comparison of the RANS and LES results it is attempted to quantify the required range of perturbations to correctly represent the model uncertainties.

[1]This material is based upon work supported by the Department of Energy [NNSA] under Award Number NA28614.

6:11PM E11.00008 Redesigning a Film-Cooled Airfoil Trailing Edge using MRI Techniques[1] ,

MICHAEL BENSON, United States Military Academy, CHRISTOPHER ELKINS, JOHN EATON, Stanford University — Trailing edges of modern gas turbine blades are film cooled through cutback slots on the airfoil pressure surface. The slots are spanwise divided, forming rectangular wall jets separated by tapered lands. The 3D wall jets mix rapidly with the mainstream flow reducing the cooling effectiveness. Experiments were conducted to document the 3D mean velocity and coolant concentration fields on a baseline configuration using Magnetic Resonance Imaging (MRI) in a water flow with Re = 110,000 based on airfoil chord length. Critical flow features causing rapid mixing were identified: a separation bubble behind the slot lip, and a pair of strong longitudinal vortices formed just downstream of the slot breakout. The geometry was modified to improve film cooling surface effectiveness obtained from the concentration field. The first redesign modified the slot lip and land shapes to minimize the slot lip separation bubble size and reduce 3D effects. The other redesigns modified the land shape to reduce the strength of the longitudinal vortices. These latter two designs produced a substantial reduction in the mixing rate of the coolant jet with the mainstream flow, improving the cooling system performance. The highly detailed concentration and velocity fields available with MRI-based experiments can be used to understand the flow physics and derive significant system improvements.

[1]This work was generously supported by GE Aviation under the GE-USA program and the Army Research Office.

6:24PM E11.00009 Three Dimensional Transient Turbulent Simulations of Scramjet Fuel Injection and Combustion ,

MARWANE BAHBAZ, University of Central Florida — Scramjet is a propulsion system that is more effective for hypersonic flights (M>5). The main objective of the simulation is to understand both the mixing and combustion process of air flow using hydrogen fuel in high speed environment s. The understanding of this phenomenon is used to determine the number of fuel injectors required to increase combustion efficiency and energy transfer. Due to the complexity of this simulation, multiple software tools are used to achieve this objective. First, Solid works is used to draw a scramjet combustor with accurate measurements. Second software tool used is Gambit; It is used to make several types of meshes for the scramjet combustor. Finally, Open Foam and CFD++ are software used to process and post process the scramjet combustor. At this stage, the simulation is divided into two categories. The cold flow category is a series of simulations that include subsonic and supersonic turbulent air flow across the combustor channel with fuel interaction from one or more injectors'. The second category is the combustion simulations which involve fluid flow and fuel mixing with ignition. The simulation and modeling of scramjet combustor will assist to investigate and understand the combustion process and energy transfer in hypersonic environment.

6:37PM E11.00010 Stochastic modeling of statistically unsteady turbulent mixing[1] ,

A.T. QAMAR, M. CADJAN, S.I. ABARZHI, The University of Chicago — Rayleigh-Taylor and Richtmyer-Meshkov turbulent mixing are statistically unsteady processes. Their dynamics combines coherence and randomness, and their mean values and fluctuations are both time-dependent. These turbulent processes have a number of symmetries and are characterized by a set of invariant measures [EPL 91, 12867]. Employing these invariant measures, we developed a stochastic model for Rayleigh-Taylor and Richtmyer-Meshkov turbulent mixing in the case of sustained, time-dependent and impulsive acceleration. For the flow quantities, the effect of fluctuations on the mean values is studied and their statistical properties are analyzed. Requirements for statistical quality of experimental and numerical data are outlined. Mechanisms of mitigation and control of turbulent mixing processes are proposed. Their implementation in experiments and simulations is discussed.

[1]The work is supported by the US National Science Foundation and by the US Department of Energy.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E12 Turbulence Simulation III 315

4:40PM E12.00001 Direct numerical simulation of stationary homogeneous stratified sheared turbulence ,

DANIEL CHUNG, GEORGIOS MATHEOU, Jet Propulsion Laboratory/California Institute of Technology — Using direct numerical simulation (DNS), we investigate stationary and homogeneous driven turbulence in various stratifications, ranging from neutral to very stable. The Taylor Reynolds number is about 400, allowing an adequate separation of scales for the study of stratified turbulence dynamics. Analysis of the simulations is used to elucidate several aspects of stratified turbulence, including flux–gradient relations, length scales, spectra, the formation of layer, and criteria of turbulence collapse.

4:53PM E12.00002 High resolution DNS studies of long-time behavior of homogeneous turbulent shear flow ,

PARVEZ SUKHESWALLA, T. VAITHIANATHAN, LANCE R. COLLINS, Cornell University — As discussed in Isaza & Collins [*J. Fluid Mech.* **678**:14–40, 2011], the shear parameter S^* has a pronounced effect on velocity gradient statistics for homogeneous turbulent shear flow (HTSF). Due to the importance of this effect, especially for higher S^*, we extended those studies to higher resolution using a new direct numerical simulation (DNS) code based on a pseudospectral algorithm that avoids remeshing [Brucker et al., *J. Comp. Phys.* **225**:20–32, 2007], and decomposes the domain into "pencils". We present DNS with $2048 \times 1024 \times 1024$ grid points, achieving a maximum Taylor microscale Reynolds number of 300. The peak in the initial energy spectrum, viscosity, and box configuration also have been optimized to maximize the time window for well-resolved simulations (up to $St = 20$), ensuring we are well into the asymptotic regime. The DNS runs confirm the sensitivity of the large- and small-scale statistics to S^*, as was found by Isaza & Collins. We also investigated the interaction between the fluctuating vorticity vector and rate-of-strain tensor as a function of scale, and find alignments vary dramatically, suggesting the primary source of enstrophy is at large scales, followed by a forward cascade to small scales. This helps explain the persistent sensitivity of the velocity gradient statistics to S^*. The combination of results suggests a new framework for modeling HTSF at high values of S^*.

5:06PM E12.00003 DNS investigation of late-stage transition in hypersonic channel flow ,

ZHIMIN XIE, SHARATH GIRIMAJI, Texas A&M University — We perform direct numerical simulations (DNS) of normal-mode evolution in hypersonic channel flows to investigate late-stage transition physics. A well-validated compressible flow solver based on Gas-Kinetic Method (GKM) is used in the computations. In this temporal DNS, periodic boundary condition is employed in the streamwise direction and wall conditions at the normal boundaries. The DNS code is first validated against analytical transition (Orr-Sommerfeld) results in the incompressible flow regime. In the compressible regime, the code is validated against homogeneous shear flow rapid distortion theory (RDT) data. Direct numerical simulation of normal modes in laminar channel flow at very high Mach number shows that the evolution exhibits a three-stage behavior similar to that observed in many hypersonic boundary layer experiments and RDT of homogeneous shear flow. The physics associated with each transition stage is investigated in great detail and a physical picture of late-stage transition is proposed.

5:19PM E12.00004 Study of high speed turbulent jets in crosslfow using numerical simulations[1]

, XIAOCHUAN CHAI, KRISHNAN MAHESH, University of Minnesota — Numerical simulations are used to study a sonic jet injected into a supersonic crossflow and a supersonic jet injected into a subsonic crossflow, where the flow conditions are based on Santiago *et al.*'s (1997) and Beresh *et al.*'s (2005) experiments, respectively. A finite volume compressible Navier–Stokes solver on unstructured grids (Park & Mahesh 2007) is used. The simulations successfully reproduce experimentally observed shock system and turbulent flow structures such as the jet shear layer vortices, wake vortices, horseshoe vortices that wrap up in front of the jet and the counter rotating vortex pair (CVP) downstream of the jet. The dynamics of these flow structures are discussed. The effects of grid resolution and crossflow boundary layer condition are studied, as well as the contribution of sub-grid scale model. The time averaged flow fields are compared to the experimental results, and reasonable agreement is observed.

[1]This work is supported by the National Science Foundation under grand CTS-0828162

5:32PM E12.00005 Lattice Boltzmann Simulations of Skin-Friction Drag Reduction in Turbulent Channel Flow with Slip/No Slip Wall Ridges

, AMIRREZA RASTEGARI, RAYHANEH AKHAVAN, University of Michigan — To gain a better understanding of the mechanisms at work in skin friction drag reduction with superhydrophobic surfaces, Lattice Boltzmann simulations were performed in turbulent channels with alternating slip/no slip ridges on the walls. Simulations were performed in turbulent channels of size $5h \times 2.5 \times 2h$ and $10h \times 5h \times 2h$ at a base Reynolds number of $Re_\tau \sim 230$. Alternating slip/no slip ridges of width $4 \leq w+ \leq 140$, aligned in the streamwise direction, all with the same fractional area of slip boundary, were studied. Drag reductions of 4%, 8%, 21%, 33% and 47%, corresponding to slip velocities of $U_{slip}/U_{bulk} =$ 0.05, 0.1, 0.26, 0.31 and 0.36 were observed for $w+ = g+ =$ 4, 8, 40, 70 and 140, respectively. The mean velocity profiles display the characteristics of combined slip described by Min and Kim [Min et al. 2004]. The streamwise and spanwise turbulence intensities show large slips at the wall, the magnitude of which increases with increasing drag reduction. Examination of the anisotropy invariant maps shows a shift of turbulence structure towards the one-dimensional turbulence limit near the wall with increasing drag reduction. For $z^+ > 25$, the turbulence structure returns to the isotropic limit.

5:45PM E12.00006 High-resolution simulations of forced compressible isotropic turbulence

, SHRIRAM JAGANNATHAN, DIEGO DONZIS, Texas A&M University — Direct numerical simulations of compressible turbulent flows are several times more expensive than their incompressible counterparts. Therefore, using large computing resources efficiently is even more pressing when studying compressible turbulence. A highly scalable code is presented which is used to perform simulations aimed at understanding fundamental turbulent processes. The code, which is based on a 2D domain decomposition, is shown to scale well up to 128k cores. To attain a statistically stationary state a new scheme is developed which involves large-scale stochastic forcing (solenoidal or dilatational) and a procedure to keep mean internal energy constant. The resulting flows show characteristics consistent with results in the literature. The attainable Reynolds and turbulent Mach numbers for given computational resources depend on the number of grid points and the degree to which the smallest scales are resolved that are given by Kolmogorov scales. A systematic comparison of simulations at different resolutions suggests that the resolution needed depends on the particular statistic being considered. The resulting database is used to investigate small-scale universality, the scaling of spectra of velocity, density and temperature fields, structure functions and the trends towards high-Reynolds number asymptotes. Differences with incompressible results are highlighted.

5:58PM E12.00007 Interaction of a converging shock wave with isotropic turbulence[1]

, ANKIT BHAGATWALA, SANJIVA LELE, Stanford University — Simulations of converging spherical shock waves propagating through a region of compressible isotropic turbulence (Taylor scale Re = 100, Mt = 0.6) are carried out. Parametric variation with respect to initial shock Mach number is studied. Both converging and reflected phases of the shock are studied. Vorticity and turbulent kinetic energy are amplified due to passage of the shock. The vorticity-dilatation term is primarily responsible for the observed behavior. This is confirmed through Eulerian and Lagrangian statistics. Transverse vorticity amplification is compared with linear planar shock turbulence theory. The smallest eddies, represented by the Kolmogorov scale, decrease in size after passing through the converging shock. Distortion of the shock due to turbulence is also investigated and quantified. Turbulence also affects maximum compression achieved at the point of shock reflection, when the shock radius is at a minimum.

[1]DOE-SciDAC project, Argonne National Labs

6:11PM E12.00008 Lagrangian motions and distribution of particles in strained turbulence

, CHUNG-MIN LEE, California State University Long Beach, PRASAD PERLEKAR, FEDERICO TOSCHI, Eindhoven University of Technology, ARMANN GYLFASON, Reykjavik University — Direct numerical simulation is employed to study the influence of straining on the motions of passive and inertial particles in turbulent flows. Our focus is on parametric dependencies of particle distribution statistics, as well as Lagrangian velocity and acceleration statistics. Results are compared with our new experimental data, as well as existing numerical and experimental data. Our numerical algorithm is based on the Rogallo method and simulates the flow field in a non-cubical, deforming domain and our particle advancing scheme assumes one-way coupling between the flow field and the particle field and is specifically adapted to the present flow geometry.

6:24PM E12.00009 Solenoidal Synthetic Turbulent Velocity Field for LES Inflow and Initial Conditions

, ADRIAN SESCU, The Johns Hopkins University, RAY HIXON, The University of Toledo, CHARLES MENEVEAU, The Johns Hopkins University — Current numerical techniques for the prediction of realistic turbulent flows require turbulent inflow or/and initial conditions that at least must match a given set of statistics and satisfy the divergence-free condition. These techniques are increasingly coupled with aeroacoustic calculations wherein the propagation of acoustic waves has to be accurately captured, as opposed to traditional CFD methods where the acoustic waves are simply damped out. This work proposes a method to generate a divergence-free turbulent velocity field based on the assumption that turbulence can be considered as a summation of random eddies satisfying specific statistics. The streamfunction concept and the requirement that the individual eddies must satisfy the linearized momentum equations about the mean flow are used to enforce the divergence-free condition. Experimental benchmark data for spatially decaying turbulence and nonhomogeneous rotor-wake turbulence from NASA's Fan Source Diagnostic Test are used to evaluate the proposed synthetic eddy method.

6:37PM E12.00010 Direct Numerical Simulation of Stationary Homogeneous Variable-Density Turbulence

, JAIYOUNG RYU, DANIEL LIVESCU, Los Alamos National Laboratory — The turbulence characteristics in statistically stationary turbulent flows composed of two incompressible miscible fluids with different densities are studied using Direct Numerical Simulations (DNS). When the two fluids have very different densities, the differential inertial effects lead to active scalar behavior. To distinguish this from the Boussinesq case, when the two fluids have commensurate densities, we call the former variable-density flows. In order to achieve a statistically stationary state, the velocity field is forced in the real space, using a linear forcing mechanism, while the active scalar field is de-mixed at a rate counteracting the normal diffusion processes using a real-space forcing based on a chemical reaction analogy. The active scalar influence on the turbulence characteristics are discussed as well as the ratios of turbulence to scalar scales for different Atwood, Schmidt, and Reynolds numbers and forcing properties. Finally, the decay of variable-density turbulence from various stationary states is also studied.

Sunday, November 20, 2011 4:40PM - 6:50PM –

Session E13 Biofluids: Cardiovascular: FSI and CFD 316

4:40PM E13.00001 Computational analysis of fluid-structure interaction in blood vessels[1] , YULIA PEET, MICHAEL MIKSIS, STEPHEN DAVIS, DAVID CHOPP, Northwestern University — Vessels carrying blood flow in a human body are known to be flexible tissues. Interaction of the internal blood flow with the vessel wall compliance, in addition to significant alteration of the fluid mechanical properties (shear and normal stresses), can also result in a variety of interesting mechanical phenomena, such as flow limitation, self-exciting oscillations (flutter), wall collapse. We investigate computationally an unsteady behavior of flexible vessels carrying a blood flow assuming incompressible newtonian fluid approximation for blood. We first utilize a membrane model with constant tension for the vessel walls and apply it to simulate the coupled fluid-wall behavior in 2D collapsible channels and 3D collapsible tubes. We show that although the model works well for 2D cases, it always leads to a complete wall collapse for 3D cases for any negative transmural pressure difference, showing the necessity of including the bending rigidity. We revisit the same problems using full linear elasticity fluid-structure interaction model for finite thickness walls developed in a high-order spectral element fluid solver. We finally investigate the unsteady flow behavior in flexible channels and tubes with the newly-developed FSI solver.

[1]We acknowledge the support of NSF RTG grant DMS-0636574.

4:53PM E13.00002 An Immersed Boundary Finite-Element Solver for Flow-Induced Deformation of Soft Structures with Application in Cardiac Flows , RAJNEESH BHARDWAJ, RAJAT MITTAL, Johns Hopkins University — The modeling of complex biological phenomena such as cardiac mechanics is challenging. It involves complex three dimensional geometries, moving structure boundaries inside the fluid domain and large flow-induced deformations of the structure. We present a fluid-structure interaction solver (FSI) which couples a sharp-interface immersed boundary method for flow simulation with a powerful finite-element based structure dynamics solver. An implicit partitioned (or segregated) approach is implemented to ensure the stability of the solver. We validate the FSI solver with published benchmark for a configuration which involves a thin elastic plate attached to a rigid cylinder. The frequency and amplitude of the oscillations of the plate are in good agreement with published results and non-linear dynamics of the plate and its coupling with the flow field are discussed. The FSI solver is used to understand left-ventricular hemodynamics and flow-induced dynamics of mitral leaflets during early diastolic filling and results from this study are presented.

5:06PM E13.00003 Fluid-structure Interaction Simulations of Deformable Soft Tissue[1] , IMAN BORAZJANI, Mechanical and Aerospace Engineering Department, University at Buffalo SUNY — Soft tissue interacts with the surrounding fluid environment in many biological and biomedical applications. Simulating such an interaction is quite challenging due to the large non-linear deformations of tissue, flow pulsatility, transition to turbulence, and non-linear fluid-structure interaction. We have extended our previous three-dimensional fluid-structure interaction (FSI) framework for rigid bodies (Borazjani, Ge, and Sotiropoulos, Journal of Computational Physics, 2008) to deformable soft tissue by coupling our incompressible Navier-Stokes solver for fluids with a non-linear large deformation finite element method for soft tissue. We use Fung-type constitutive law for the soft tissue that can capture the stress-strain behavior of the tissue. The FSI solver adopts a strongly-coupled partitioned approach that is stabilized with under-relaxation and Aitken acceleration technique. We validate our solvers against the experimental data for tissue valves and elastic tubes. We show the capabilities of our solver by simulating the fluid-structure interaction of tissue valves implanted in the aortic positions and elastic collapsible tubes.

[1]This work was partly supported by the Center for Computational Research at the University at Buffalo.

5:19PM E13.00004 Towards a Fast Dynamic Model of the Human Circulatory System , CAROLYN KAPLAN, MELISSA GREEN, JAY BORIS, ELAINE ORAN, Naval Research Laboratory — We describe a model for systems-level transport in the human circulatory system that is based on a set of equations for a one-dimensional unsteady elastic pipe flow circuit. The system is collapsed from three spatial dimensions and time to one spatial dimension and time by assuming axisymmetric vessel geometry and a parabolic velocity profile across the cylindrical vessels. To drive the fluid, the contractions of a beating heart are modeled as periodic area changes of the elastic vessels. Two different models are compared, both including and neglecting fluid acceleration. Time-resolved distributions of pressure, velocity and area compare reasonably well with reference data. Increasing the rigidity of the vasculature is found to increase peak arterial pressures on the order of ten percent, and including a distributed vascular contraction to model distributed skeletal muscle contractions monotonically increases time-averaged blood flow in the veins, consistent with human physiological response. The circulatory system model presented here simulates the circulatory system on the order of one hundred times faster than real-time; that is, we can compute thousands of heartbeats per minute of CPU time.

5:32PM E13.00005 A comparison between Dirichlet and Neumann boundary conditions for 0D/3D coupling in cardiovascular simulations , MAHDI ESMAILY MOGHADAM, MAE Dept., UC San Diego, TAIN-YEN HSIA, Great Ormond Street Hospital, UK, ALISON MARSDEN, MAE Dept., UC San Diego — Implementation of boundary conditions (BCs) in cardiovascular simulations poses numerical challenges due to the complex dynamic behavior of the circulatory system. A closed-loop lumped parameter network (LPN) coupled to a 3D domain is a powerful tool that can be used to model the global dynamics of the circulatory system and its response to local changes in surgery design. In this study, the essential formulations for coupling a 0D model using both Dirichlet and Neumann BCs to a discretized 3D finite element domain are discussed. Using a closed loop LPN with a heart model and pure Dirichlet or Neumann BCs, the limitations of these two approaches are studied and stability, accuracy, and computational cost are compared. Results show that the Dirichlet BC is more accurate for the tested mesh sizes with better stability characteristic at larger time step sizes, although this method requires additional velocity profile information. Application to patient specific models is also presented and discussed.

5:45PM E13.00006 Resolving Low-Density Lipoprotein (LDL) on the Human Aortic Surface Using Large Eddy Simulation , JONAS LANTZ, MATTS KARLSSON, Linköping University, Sweden — The prediction and understanding of the genesis of vascular diseases is one of the grand challenges in biofluid engineering. The progression of atherosclerosis is correlated to the build- up of LDL on the arterial surface, which is affected by the blood flow. A multi-physics simulation of LDL mass transport in the blood and through the arterial wall of a subject specific human aorta was performed, employing a LES turbulence model to resolve the turbulent flow. Geometry and velocity measurements from magnetic resonance imaging (MRI) were incorporated to assure physiological relevance of the simulation. Due to the turbulent nature of the flow, consecutive cardiac cycles are not identical, neither in vivo nor in the simulations. A phase average based on a large number of cardiac cycles is therefore computed, which is the proper way to get reliable statistical results from a LES simulation. In total, 50 cardiac cycles were simulated, yielding over 2.5 Billion data points to be post-processed. An inverse relation between LDL and WSS was found; LDL accumulated on locations where WSS was low and vice-versa. Large temporal differences were present, with the concentration level decreasing during systolic acceleration and increasing during the deceleration phase. This method makes it possible to resolve the localization of LDL accumulation in the normal human aorta with its complex transitional flow.

5:58PM E13.00007 Computational Fluid Dynamics Analysis of Thoracic Aortic Dissection , YIK SAU TANG, YI FAN, Department of Mechanical Engineering, The University of Hong Kong, STEPHEN WING KEUNG CHENG, Department of Surgery, The University of Hong Kong, KWOK WING CHOW, Department of Mechanical Engineering, The University of Hong Kong — Thoracic Aortic Dissection (TAD) is a cardiovascular disease with high mortality. An aortic dissection is formed when blood infiltrates the layers of the vascular wall, and a new artificial channel, the false lumen, is created. The expansion of the blood vessel due to the weakened wall enhances the risk of rupture. Computational fluid dynamics analysis is performed to study the hemodynamics of this pathological condition. Both idealized geometry and realistic patient configurations from computed tomography (CT) images are investigated. Physiological boundary conditions from in vivo measurements are employed. Flow configuration and biomechanical forces are studied. Quantitative analysis allows clinicians to assess the risk of rupture in making decision regarding surgical intervention.

6:11PM E13.00008 Investigation of geometry effects on blood flow in flexible carotid artery bifurcation[1] , SANG HOON LEE, Seoul National University, HYOUNG GWON CHOI, Seoul National University of Science and Technology, JUNG YUL YOO, Seoul National University — To investigate the effect of the flexible artery wall on the flow field, numerical simulations for the blood flow are carried out. For solving the equation of motion for the structure in fluid-structure interaction problems, it is necessary to calculate the fluid force on the surface of the structure explicitly. To avoid the complexity due to the necessity of additional mechanical constraints, we use the combined formulation which includes both the fluid and structural equations of motion into single coupled variational equation. The Navier-Stokes equations for fluid flow are solved using a P2P1 Galerkin finite element method and mesh movement is achieved using arbitrary Lagrangian-Eulerian formulation. The Newmark method is employed for solving the dynamic equilibrium equations for linear elastic solid mechanics. The pulsatile, three-dimensional, incompressible flows of Newtonian fluids constrained in the flexible wall are analyzed. The study shows that flexibility of carotid wall affects significantly the flow phenomena during the pulse cycle. It is found that the flow field is also strongly influenced by bifurcation angle.

[1]This work was supported by Mid-career Researcher Program through the NRF funded by the MEST (2009-0079936) and KISTI supercomputing center (KSC-2010-C1-0025).

6:24PM E13.00009 Optimum Heart Rate to Minimize Pulsatile External Cardiac Power , NIEMA PAHLEVAN, MORTEZA GHARIB, California Institute of Technology — The workload on the left ventricle is composed of steady and pulsatile components. Clinical investigations have confirmed that an abnormal pulsatile load plays an important role in the pathogenesis of left ventricular hypertrophy (LVH) and progression of LVH to congestive heart failure (CHF). The pulsatile load is the result of the complex dynamics of wave propagation and reflection in the compliant arterial vasculature. We hypothesize that aortic waves can be optimized to reduce the left ventricular (LV) pulsatile load. We used an in-vitro experimental approach to investigate our hypothesis. A unique hydraulic model was used for in-vitro experiments. This model has physical and dynamical properties similar to the heart-aorta system. Different compliant models of the artificial aorta were used to test the hypothesis under various aortic rigidities. Our results indicate that: i) there is an optimum heart rate that minimizes LV pulsatile power (this is in agreement with our previous computational study); ii) introducing an extra reflection site at the specific location along the aorta creates constructive wave conditions that reduce the LV pulsatile power.

6:37PM E13.00010 Intraventricular vorticity favors conservation of kinetic energy along the cardiac cycle: analysis in patients with dilated cardiomyopathy by post-processing color-doppler images[1] , MARTA ALHAMA, YOLANDA BENITO, JAVIER BERMEJO, RAQUEL YOTTI, ESTHER PEREZ-DAVID, ALICIA BARRIO, CANDELAS PEREZ DEL VILLAR, ANA GONZALEZ MANSILLA, FRANCISCO FERNANDEZ AVILES, Division of Cardiology, Hospital General Universitario Gregorio Maranon, Madrid, JUAN CARLOS DEL ALAMO, MAE Department, UC San Diego — Background: This study assesses if the left ventricle (LV) filling vortex developed during diastole may be a mechanism that improves systolic efficiency. 19 patients with dilated cardiomyopathy (DCM) and 37 healthy volunteers were studied. Recently, we have developed and validated a method that derives two-dimensional maps of the LV flow from standard color-Doppler sequences. Two-dimensional maps of instantaneous LV flow were obtained, and circulation, energy and position of the main and secondary vortices were calculated along the cardiac cycle. At aortic valve opening (AVO) the vortex circulation is higher in DCM subjects than healthy volunteers. However, the position of the vortex is farthest form LV outflow tract (LVOT), and this results in lower flow velocity in LVOT at AVO. This phenomenon is altered in patients with DCM.

[1]Supported by ISCIII (Spain) and NIH 1 R21 HL108268-01 (US).

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E14 Biofluids: Cellular II: Blood Transport 317

4:40PM E14.00001 ABSTRACT WITHDRAWN –

4:53PM E14.00002 Quantifying the glycocalyx effects in blood flow in capillaries , MINGGE DENG, HUAN LEI, BRUCE CASWELL, GEORGE KARNIADAKIS, Brown University — We employ Dissipative Particle Dynamics (DPD) to simulate blood flow in small capillaries with the glycocalyx attached to the endothelial surface. The effects of the glycocalyx on hematocrit and resistance to blood flow are analyzed by comparing with and without glycocalyx attached to the surface. Of particular interest is the quantification of the slip boundary condition at the edge of glycocalyx and also of the glycocalyx deformation at different grafting densities, stiffness and height of the glycocalyx. In addition to the physical insight gained for this important but relatively unexplored bio-flow, simple models for the slip velocity will be proposed that can be used in continuum simulations of blood flow in micro-vessels.

5:06PM E14.00003 Mechanism of flow induced segregation in suspensions of binary mixtures of deformable capsules based on rigidity in confined geometries , AMIT KUMAR, MICHAEL GRAHAM, University of Wisconsin - Madison — Flow induced segregation in mixtures of deformable particles based on rigidity is relevant in many biological and technological applications. For example, this property can be employed in the detection or separation of stiffened RBCs in various diseased states like malaria in a point of care microfluidic device. We numerically study here neo-Hookean capsule suspensions subjected to pressure driven flows in a slit geometry using an accelerated implementation of the boundary integral method. The effect of a wide variety of parameters like volume fraction, capillary number, confinement ratio, number fraction of the floppy particle (X), and the rigidity ratio between the two components of the mixture were explored. In pure suspensions, the mean wall normal position of the stiff and the floppy particles were comparable; however, in mixtures, the stiff particles were found to be increasingly displaced towards the walls with increasing X. Simple model studies involving pair collisions between stiff and floppy particles qualitatively explain the above behavior. Numerical results are further incorporated in the Fokker-Planck equation, which is found to correctly model the particle motion and predict the segregation behavior. The drift and diffusion terms in the Fokker-Planck equation are consistent with the results of pair collision.

5:19PM E14.00004 Simulation of neutrophil motion and deformation: Influence of rheology and flow configuration , MELANIE LE ROUX, INP IMFT Toulouse, JACQUES MAGNAUDET — We report on preliminary computations of neutrophils (white cells) flowing and deforming in capillary vessels in different flow configurations. We crudely consider that the cell is made of a solid core and a viscoelastic cytoplasm surrounded by an elastic membrane and moves within a Newtonian plasma. A Volume Of Fluid approach is employed to follow the displacement of the cell while the Immersed Boundary Method is used to take into account the presence of a solid core; the viscoelastic behavior of the cytoplasm is mimicked using the Oldroyd-B model. We discuss the behavior of the cell when placed in 3 different flow configurations, namely a four roll-mill device allowing all types of linear flow fields to be generated in the central region where the cell stands, a contraction followed by an expansion in a straight channel, and a periodic network of prismatic blocks. The dimensions of the last two systems are such that the cell experiences large deformations while moving. We discuss the influence of the physical parameters of the system, especially the Capillary and Deborah numbers and the ratio of the inner and outer Newtonian viscosities, on the flow and cell evolution.

5:32PM E14.00005 Multiscale modeling of sickle anemia blood blow by Dissipative Partice Dynamics , HUAN LEI, BRUCE CASWELL, GEORGE KARNIADAKIS, Brown University — A multi-scale model for sickle red blood cell is developed based on Dissipative Particle Dynamics (DPD). Different cell morphologies (sickle, granular, elongated shapes) typically observed in *in vitro* and *in vivo* are constructed and the deviations from the biconcave shape is quantified by the Asphericity and Elliptical shape factors. The rheology of sickle blood is studied in both shear and pipe flow systems. The flow resistance obtained from both systems exhibits a larger value than the healthy blood flow due to the abnormal cell properties. However, the vaso-occulusion phenomenon, reported in a recent microfluid experiment, is not observed in the pipe flow system unless the adhesive interactions between sickle blood cells and endothelium properly introduced into the model.

5:45PM E14.00006 Tracer-incorporated X-ray imaging of biofluid flow phenomena , SUNG YONG JUNG, SUNGSOOK AHN, SANG JOON LEE, POSTECH, POSTECH TEAM — Particle-traced X-ray imaging technologies have been developed by combining the merits of the X-ray radiography and particle image velocimetry (PIV) technique. The developed X-ray imaging technology has strong potential in the noninvasive analysis of various flows such as non-transparent fluid flows or fluids flowing in opaque conduits. In this study, tracer-incorporated X-ray imaging technology was developed. In addition, new- concepted tracer particles were designed for in vitro and in vivo X-ray imaging analysis of various biofluids. As tracer particles in X-ray image, X-ray contrast enhancer Iopamidol was encapsulated into bio-compatible polymeric chitosan microparticles and gold nanoparticles with high X-ray absorption efficiency were directly incorporated into cells. The Iopamidol-incorporated polymeric microparticles were successfully applied for in vivo blood flow measurement in a rat. The gold nanoparticles were selectively incorporated into cancer cells, by which cancer cells can be detected in situ. The developed X-ray imaging technology would have a great potential in biomedical applications such as in situ analysis of blood flow and cancer detection.

5:58PM E14.00007 Transport of magnetic nanobeads in a small blood vessel , JONATHAN FREUND, University of Illinois at Urbana-Champaign — Numerical simulations are used to study the transport of sub-micron spherical magnetic beads in a model microvessel, particularly how transport is affected by physiologically realistic concentrations of flowing red cells. (Such beads are candidate vehicles for targeted drug delivery.) A previously validated high-fidelity boundary integral algorithm is used to solve the flow equations in the viscous limit in a $\sim 15\mu$m vessel. As expected, the red cells suppress transport toward the vessel wall when a magnetic field is applied normal to the flow direction. A more subtle mechanism is important when a magnetic field has a component parallel to the vessel. A homogeneous fluid model would be insensitive to forces applied in this direction, but tendency of blood cells to tilt away from the wall on their upstream side breaks the streamwise symmetry. These tilted cells act as viscous-flow variants of turning vanes, guiding magnets toward the wall if they are slowed by the magnetic field and toward the vessel center if they are accelerated by the magnetic field. This affects dispersion in the vessel and can also alter wall-ward magnetic field driven transport.

6:11PM E14.00008 White blood cell deformation and firm adhesion , ALEX SZATMARY, CHARLES EGGLETON, University of Maryland, Baltimore County — For a white blood cell (WBC) to arrive at infection sites, it forms chemical attachments with activated endothelial cells. First, it bonds with P-selectin, which holds it to the wall, but weakly; this allows the WBC to roll under the shear flow of the blood around it. Later, the WBCs bond with the stronger intracellular adhesion molecule-1 (ICAM-1); it is these ICAM bonds that allow the WBCs to fully resist the flow and stop rolling, allowing them to crawl through the endothelial wall. We model this numerically. Our model uses the immersed boundary method to represent the interaction of the shear flow with the deformable cell membrane. Receptors are on the tips of microvilli-little fingers sticking off of the cell membrane. The microvilli also deform. The receptors stochastically form and break bonds with molecules on the wall. Using this method, the history of each microvillus and its bonds can be found, as well as the distribution of the adhesion traction forces and how all of these vary with the deformability of the white blood cell. At higher shear rates, the white blood cell membrane deforms more, increasing its contact area with the surface; this effect is larger for softer membranes. We investigate how the deformability of the WBC affects the ease with which it forms firm adhesion.

6:24PM E14.00009 Modification of Platelet Margination Rate via Reduction of Viscosity Ratio[1] , DANIEL REASOR, MARMAR MEHRABADI, DAVID KU, CYRUS AIDUN, Georgia Institute of Technology — Experimental investigations of platelet margination have primarily been limited to effects of hematocrit (Ht.) and shear rate. The suspending fluids used commonly have viscosities greater than plasma which can modify the transition in dynamical regimes from tumbling to tank-treading for isolated RBCs. This work focuses on the effects of λ, the ratio of internal to suspending fluid viscosity of RBCs, on the rate of platelet margination in a rigid 41.3 μm diameter vessel. Simulations are performed with a lattice-Boltzmann fluid solver using the standard bounce-back boundary condition coupled with a coarse-grained spectrin-link RBC membrane model and a Newtonian dynamics solver for rigid platelets. Our results are consistent with observations that an increase in Ht. increases the rate of platelet margination for Ht.=20-40%, but we focus on the modification of λ at Ht.=20%. Our results show that rigid RBCs inhibit margination, but modifying λ with deformable RBCs show significant increases in margination rate. Our observations demonstrate an increase in platelet wall-normal velocity fluctuations, enhanced margination rate, and an increase in the wall-normal diffusivity as λ is reduced from the physiological value of five.

[1]NSF TeraGrid Grant: TG-CTS100012

6:37PM E14.00010 The Effect of Discoid Shape on Platelet Margination in a Microvessel[1] , MARMAR MEHRABADI, DANIEL REASOR, DAVID KU, CYRUS AIDUN, Georgia Institute of Technology — Margination of platelets to the skimming layer only occurs above a threshold hematocrit (Ht.). Platelet size and concentration in blood is much smaller compared to red blood cells (RBCs). We study the hypothesis that platelets are passively convected to vessel walls and that their morphology can effect margination rate and dynamics. To examine this, we study the influence of particle shape on margination rate through changing the aspect ratio (AR) of rigid particles at fixed volume. We use a coarse-grained spectrin-link method for RBC membranes and Newtonian dynamics for rigid particles coupled with a 3D lattice-Boltzmann fluid solver using standard bounce-back boundary conditions. Simulations are performed at Ht.=20% in a $41.\bar{3}$ μm vessel. Our results show that AR has a significant effect on margination rate: Lower AR particles marginate more rapidly. We also show that the higher AR particles "flip" and "slide" between RBCs as they migrate to the skimming layer. The final location of particles in the skimming layer is also influenced by their shape. Thin disk-shaped particles interact with the RBCs at the edge of the skimming layer more frequently than with the vessel wall while spherical particles interact with both simultaneously.

[1]NSF TeraGrid Grant: TG-CTS100012

Sunday, November 20, 2011 4:40PM - 6:37PM –
Session E15 Porous Media I 318

4:40PM E15.00001 Vapor transport through short hydrophobic nanopores for desalination[1] , JONGHO LEE, SEAN O'HERN, Massachusetts Institute of Technology, TAHAR LAOUI, FAIZUR RAHMAN, King Fahd University of Petroleum and Minerals, ROHIT KARNIK, Massachusetts Institute of Technology — We propose a concept for desalination of water by reverse osmosis (RO) using a vapor-trapping membrane composed of short hydrophobic nanopores and separates the salt water (feed) and the fresh water (permeate) on each side. The feed water is vaporized by applied pressure and the water vapor condenses on the permeate side accompanied by recovery of latent heat. A probabilistic model based on rarified gas conditions predicted 3-5 times larger mass flux by the proposed membrane than conventional RO membranes at temperatures in the range of 30-50C. To realize the short hydrophobic nanopores, gold was deposited at the entrance of alumina pores followed by SAM formation. The fraction of leaking pores was confirmed to be less than 0.2% using a calcium ion indicator (Fluo-4). Finally, a microfluidic flow cell was fabricated for characterizing the transport properties of the membranes.

[1]The authors would like to thank the King Fahd University of Petroleum and Minerals in Dhahran, Saudi Arabia, for funding the research reported in this paper through the Center for Clean Water and Clean Energy at MIT and KFUPM.

4:53PM E15.00002 Wrinkling of wet paper[1] , HO-YOUNG KIM, JUNGCHUL KIM, Seoul National University, L. MAHADEVAN, Harvard University — It is a mundane experience that paper stained with water wrinkles. It is because a wetted portion of paper, which swells due to the hygroexpansive nature of the cellulose fiber network, deforms out of its original plane. Here we quantify the dynamics of wrinkling of wet paper coupled to the capillary imbibition of water into paper using a combination of experiment and theory. While supplying water from a capillary tube that touches the center of a paper strip, we measure the spreading rate of the wet area, wait time for the out-of-plane buckling, and temporal growth of a wrinkling magnitude. Using a theoretical model assuming a linear increase of the strain and an exponential decay of the elastic modulus with the water concentration, we construct scaling laws to predict the simultaneous capillary imbibition and wrinkling rates.

[1]This work was supported by the Wyss Institute of Harvard University.

5:06PM E15.00003 Permeability reduction of self-affine fractures explained by means of the critical barrier concept , LAURENT TALON, HAROLD AURADOU, lab. FAST, CNRS, ALEX HANSEN, Norwegian University of Science and Technology — In many low permeability geological formations, flow occurs primarily through fracture networks. There is therfore a need for reliable modeling of the hydromechanical behavior of fracture. We consider here fracture with self-affine correlation. Most of the models fails to predict the effective permeability of such fracture as soon as some contact area are present. We introduce a model based on the generalization of the concept of the bottle neck which allows the prediction of the permeability of self-affine rough channels (one-dimensional fracture) and two-dimensional fractures over the entire range of possible apertures. In one-dimensional rough fracture, when the two wall are brought to contact, the permeability is increasingly controlled by the region of minimum aperture. This is the bottle neck concept. In two-dimensionnal fracture, the position of the minimum aperture is not so crucial since the flow can easily by-pass regions of low permeability. To generalize this concept, we introduce the most restrictive barrier path defined as being the barrier that has the smallest average permeability. Using numerical simulation, we identify three permeability scaling regime that will be explained by the introduction of other critical barrier ordered by its criticality.

5:19PM E15.00004 Fluid Flow in Porous Media for Soil-Water Retention , CESARE MIKHAIL CEJAS, BERTRAND SELVA, RAPHAEL BEAUFRET, LARRY HOUGH, CNRS COMPASS UMI 3254, CHRISTIAN FRETIGNY, ESPCI, REMI DREYFUS, CNRS COMPASS UMI 3254, CNRS / RHODIA / UPENN UMI 3254 TEAM — The study aims to understand the mechanisms that determine the behavior of water in soil. In developing a better comprehension of the coupling between the various fluxes (e.g. evaporation, drainage) in soil and the surrounding environment, we elaborate strategies that permit to understand and improve particularly the water absorption by the roots. Our first approach, through direct visualization, focuses on evaporation out of a 2D model soil consisting of monolayer glass beads. Evaporation from porous media exhibits an abrupt transition from capillary-supported regime 1 to diffusion-controlled regime 2. Varying the wettability of the model soil suggests that the duration of regime 1evaporation and drying front formation in hydrophobic media are shorter than in hydrophilic media due to the absence of hydraulic continuity towards the evaporating surface. We then study how evaporation couples in the presence of roots in the model soil while being subjected to various treatment conditions (e.g. physical additives, etc.). Through this study, we would be able to quantify how the physico-chemical soil treatments affect these phenomena and inspire solutions for improving soil water retention.

5:32PM E15.00005 Impact of wettability correlations on multiphase flow through porous media , MARTA SANCHEZ DE LA LAMA, MARTIN BRINKMANN, STEPHAN HERMINGHAUS, Max-Planck Institute Dynamics-Self Organization (MPI-DS), 37073 Goettingen (Germany) — In the last decades, significant progress has been made toward understanding the multiphase displacement through porous media and the role of substrate properties like homogeneous wettability or pore geometry. However, the effect of heterogeneous wettability at microscopic scales and its relation to large-scale properties, like relative permeability or capillary pressure, remains still little understood. In the present study forced imbibition through a two-dimensional porous medium is simulated at the pore scale by means of a mesoscopic particle approach [1,2]. The substrate is described as an assembly of non-overlapping circular disks whose preferential wettability is distributed according to prescribed correlations, i.e., from pore scale in terms of Janus beads up to domains at system scale. We analyze how this well-defined heterogeneous wettability affects the dynamics and try to establish a relationship among wettability-correlations and large-scales properties of the multiphase flow. References [1] Y. Inoue et al., J. Comp. Phys. 201, 191 (2004) [2] G. Gompper et al., Adv. Polym. Sci. 221, 1 (2009)

5:45PM E15.00006 Analytical model of wetting liquid unsaturated climb in a porous medium , B. MARKICEVIC, Kettering University, B. BIJELJIC, Imperial College, H.K. NAVAZ, Kettering University — The analysis of the dynamics and stability of the wetting liquid capillary climb flow in the porous medium, which is opposed by gravity force, suggests that there may be a unique analytical correlation between the capillary and Bond number given in the exponential form. Starting from this exponential expression, the analytical model for the climbing height as a function of time is obtained. The model accounts for two-step climbing dynamics, where for smaller climbing height, the height of single-phase displacement flow is identified. For larger climbing heights closer to the point in which the capillary and gravity forces equilibrate, the height of the developing multiphase flow front and interface position between partially wet and dry fraction of porous medium is defined. It turns out that the analytical solution predicts the climbing height for short and longer times. To use this approach, one needs the capillary versus Bond number correlation only, which is completely measureable from the experiments. In calculating both numbers, the average pore radius is used as the geometrical scale. There is an excellent agreement between the analytical model predictions and the set of experimental results in which the permeability varies for two orders of magnitude, and where both single-phase and multiphase climbing dynamics is observed. Finally, the generality of analytical model needs to be examined further.

5:58PM E15.00007 Effect of hydrophobicity on the flow through a porous media , O. CHAVEZ, R. ZENIT, Universidad Nacional Autonoma de Mexico, D. CHEHATA, Tecnología Aplicada en Exploración y Producción Petrolera, Inc. Mexico — We have experimentally studied the effect of hydrophobic conditions on the flow in a porous medium; our motivation arises from the flow of petroleum at well conditions, where the wettability can change drastically. A porous media made with glass beads was considered as reference to obtain the permeability. Subsequently the glass beads were coated with a hydrophobic agent in order to observe the effect on the coefficient of permeability of the porous medium, in the Darcy flow regime. Many experiments were conducted considering mixtures of sizes of beads and wettability conditions. Preliminary results will be presented and discussed. As expected, variations of the wettability of the grains affect the permeability in a significant manner.

6:11PM E15.00008 Capillary effects in buoyancy-driven spreading within a porous medium ,
JEROME NEUFELD, BP Institute, University of Cambridge, OLIVIER DUBOURDIEU, Ecole Normale Superieure de Lyon, HERBERT HUPPERT, Institute of Theoretical Geophysics, University of Cambridge — Patterns driven by the imbibition and spread of fluids in porous media due to gravity and capillarity can be found in a host of industrial and environmental settings. We show experimentally that for small, constant fluid volumes, gravity and capillarity can lead to static fluid configurations. In contrast, when the fluid exceeds a critical depth it can flow under gravity before finding a new static configuration. We show that the evolution can be modeled as a gravity current in which capillarity, manifest as hysteresis between advancing and receding interfaces, plays a key role. Model predictions of the ultimate extent of such currents are compared to our experimental results.

6:24PM E15.00009 Stable and unstable waves in two-phase porous media flow ,
KIMBERLY SPAYD, MICHAEL SHEARER, ZHENGZHENG HU, North Carolina State University — Plane waves for two phase flow in a porous medium are modeled by the one-dimensional Buckley-Leverett equation, a *scalar* conservation law. We analyze stability of sharp planar interfaces to two-dimensional perturbations, which involves a *system* of partial differential equations. Linear stability analysis, in a more general regime than the classical Saffman-Taylor analysis, results in a description of the dispersion relation to leading order in the wave number, providing a criterion that distinguishes between interfaces that are long-wave stable and those that are not. Numerical simulations of the full nonlinear system of equations, including dissipation and dispersion, illustrate the analytical results.

Sunday, November 20, 2011 4:40PM - 6:11PM –

Session E16 Computational and Theoretical Aerodynamics 319

4:40PM E16.00001 Numerical Study for Detailed Flow Fields and Performance of the Savonius-Type Rotor ,
TONG ZHOU, DIETMAR REMPFER, IIT, Chicago — The Savonius-type rotor is simple in structure, has good starting characteristics, relatively low operating speeds, and an ability to accept wind from any direction, although it has a lower efficiency than other vertical axis wind turbines. So far a number of experimental investigations have been carried out to study the performance of the Savonius rotor, however, there is a lack of detailed descriptions of the flow field. The aim of this paper is to numerically explore the non-linear two- dimensional unsteady flow over a Savonius rotor and develop a simulation method for predicting its aerodynamic performance. The simulations are based on Star CCM+. The motion of the blades is solved by using a moving mesh. Different turbulence models are compared. Parameters such as mesh density, wall y^+, and boundary conditions will be discussed. Numerical simulation results are compared with experimental data. Separation of the flow at the blade tips is well modeled. The characteristics of flow fields details are studied, including boundary layer, moment coefficient, and pressure distribution. The wall shear on each surface of the blades is studied to look into the position of the separation point. Computational fluid dynamics is proven to be an effective approach for the investigation of the Savonius-type rotor, on the premise of proper theory and reasonable assumption. It also provides a basis for optimization of the Savonius wind turbine.

4:53PM E16.00002 The Kutta-Zhukovsky Lift Theorem revisited: Alteration due to the Viscous Wake ,
SVEN SCHMITZ, The Pennsylvania State University — The circulation theory of lift comprised in the classical Kutta-Zhukovsky Lift Theorem forms the foundation of modern aerodynamic wing theory. The theorem has been applied ever since in lifting-line models of aircraft and rotary wings. Reynolds numbers larger than one million support its validity, yet the effect of a viscous wake on a change in the functional relationship between lift and circulation is not taken into account in standard lifting-line analyses. A discrepancy in circulation of more than six percent in comparison to the classical Kutta-Zhukovsky Lift Theorem has been demonstrated by the author (Schmitz & Chattot, *Computers & Fluids*, **36**) for moderately separated flow around a wind turbine airfoil by means of a control volume analysis governed by the Navier-Stokes equations. The present work extends the previous analysis to general three-dimensional flow around a lifting body. An analytical expression is presented that extends the classical Kutta-Zhukovsky Lift Theorem by adding terms to the theorem due to chord- and spanwise vorticity transport. An integrated solution for induced drag is given that has not been documented in previous literature on the subject. The generalized theorem will find future application and quantification in actuator-line methods used to predict wind farm wake interactions with Atmospheric Boundary Layer flow.

5:06PM E16.00003 A Computational Parametric Study of Drag Reducing Riblet Geometries
, AARON BOOMSMA, FOTIS SOTIROPOULOS, Univ. of Minnesota — Riblets are micro-grooved films that passively affect the skin friction of a turbulent boundary layer. Many researchers have shown that riblets can augment or decrease drag. This work utilizes high-resolution direct numerical simulations at low Reynolds numbers to conduct a parametric study of those riblets that decrease drag. Insights on drag reduction due to geometry are revealed for a variety of riblet heights, spacings, and shapes. Finally, this work discusses flow physics of the modified turbulent boundary layer and mechanisms of drag reduction. This work was supported by the Department of Energy (DE-EE0002980) and the University of Minnesota Initiative for Renewable Energy and the Environment. Computational resources were provided by the University of Minnesota Supercomputing Institute.

5:19PM E16.00004 Stability Analysis of a mortar cover ejected at various Mach numbers and angles of attack[1] ,
JANE SCHWAB, MARIA-ISABEL CARNASCIALI, JOE ANDREJCZYK, University of New Haven, MIKE KANDIS, Pioneer Aerospace — This study utilized CFD software to predict the aerodynamic coefficient of a wedge-shaped mortar cover which is ejected from a spacecraft upon deployment of its Parachute Recovery System (PRS). Concern over recontact or collision between the mortar cover and spacecraft served as the impetus for this study in which drag and moment coefficients were determined at Mach numbers from 0.3 to 1.6 at 30-degree increments. These CFD predictions were then used as inputs to a two-dimensional, multi-body, three-DoF trajectory model to calculate the relative motion of the mortar cover and spacecraft. Based upon those simulations, the study concluded a minimal/zero risk of collision with either the spacecraft or PRS.

[1]Sponsored by Pioneer Aerospace

5:32PM E16.00005 Vorticity forces on an impulsively started finite plate[1] ,
CHIN-CHOU CHU, CHENG-TA HSIEH, JIAN-JHIH LEE, CHIEN CHENG CHANG, National Taiwan University — In this talk, various force contributions to an impulsively started finite plate with a high and low aspect ratio are investigated from the perspective of a diagnostic vorticity force theory. In contrasted to the traditional pressure force analysis (PFA), the vorticity force analysis (VFA) reveals new salient features in its applications to three-dimensional flow by examining section-wise force contributions along the spanwise direction. At a large aspect ratio (AR=3), the force distributions of PFA and VFA show close agreements with each other in middle sections, while at a lower aspect ratio (AR=1), the force distribution of PFA is substantially larger than that of VFA in most of the sections. The difference is compensated by the contributions partly by the edge sections and mainly by the outer tip vortices. The present force-element analysis provides a better perspective for flow control by relating the forces directly to the various sources of vorticity (or vortex structures) in the flow.

[1]The work was supported in part by the National Science Council of the Republic of China (Taiwan) under the Contract No. NSC97-2221-E-002-223-MY3 and No. 98-2221-E-002-094-MY3.

5:45PM E16.00006 Numerical simulation of Reynolds number effects on velocity shear flow around a circular cylinder , SHUYANG CAO — Three-dimensional Direct Numerical Simulation and Large Eddy Simulation are performed to investigate the shear effects on flow around a circular cylinder at Reynolds numbers of Re=60–1000. The shear parameter, which is based on the velocity gradient, cylinder diameter and upstream mean velocity at the center plane of the cylinder, varies from 0 to 0.30. Variations of Strouhal number, drag and lift coefficients, and unsteady wake structures with shear parameter are studied, along with their dependence on Reynolds number. The presented simulation provides detailed information for the flow field around a circular cylinder in shear flow. This study shows that the Strouhal number exhibits no significant variation with shear parameter. The stagnation point moves to the high-velocity side almost linearly with shear parameter, and this result mainly influences the aerodynamic forces acting on a circular cylinder in shear flow. Both the Reynolds number and shear parameter influence the movement of the stagnation point and separation point. Mode A wake instability is suppressed into parallel vortex shedding mode at a certain shear parameter. The lift force increases with increasing shear parameter and acts from the high-velocity side to the low-velocity side.

5:58PM E16.00007 Large-Scale Numerical Simulation of Fluid Structure Interactions in Low Reynolds Number Flows , ALI EKEN, MEHMET SAHIN, Istanbul Technical University — A fully coupled numerical algorithm has been developed for the numerical simulation of large-scale fluid structure interaction problems. The incompressible Navier-Stokes equations are discretized using an Arbitrary Lagrangian-Eulerian (ALE) formulation based on the side-centered unstructured finite volume method. A special attention is given to satisfy the discrete continuity equation within each element at discrete level as well as the Geometric Conservation Law (GCL). The linear elasticity equations are discretized within the structure domain using the Galerkin finite element method. The resulting algebraic linear equations are solved in a fully coupled form using a monolitic multigrid method. The implementation of the fully coupled iterative solvers is based on the PETSc library for improving the efficiency of the parallel code. The present numerical algorithm is initially validated for a beam in cross flow and then it is used to simulate the fluid structure interaction of a membrane-wing micro aerial vehicle (MAV).

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E17 Viscous Flows 320

4:40PM E17.00001 The flow field of a submerged viscous impinging jet , J. TILAK RATNANATHER, JUNG H. KIM, Johns Hopkins University, ANTHONY M.J. DAVIS, University of California, San Diego — The flow field of a submerged viscous jet impinging on an infinite plane wall is studied. The whole creeping flow of semi-infinite extent is generated via distributions on a cylindrical pipe of tangentially and normally directed Stokeslets which are modified in two stages to achieve no-slip at the wall at $z = 0$. First the pressure and vorticity jumps associated with the Poiseuille flow upstream in the pipe are forced and then further distributions far upstream of the orifice at $z = h$ are added to achieve no slip on the pipe wall. The adjustment of the interior pipe flow from its upstream parabolic profile to its exit profile is thus included in this creeping flow analysis. The generated flow field for $h = 1, 2, 3, 4, 5$ is used to examine the behaviour of the wall pressure and wall shear stress with respect to h. This behaviour is discussed briefly in the context of physiological experiments that motivated this study.

4:53PM E17.00002 Bending of Elastic Fibers in Viscous Flow: the Influence of Confinement[1] , JASON WEXLER, Princeton Unversity, PMMH-ESPCI, HELENE BERTHET, NAWAL QUENNOUZ, OLIVIA DU ROURE, PMMH-ESPCI, HOWARD STONE, Princeton University, ANKE LINDNER, PMMH-ESPCI — Applications such as microfluidic flow sensors or living micro-organisms often involve the deformation of a slender deformable body attached to a rigid boundary. Here we investigate the deformation of an anchored elastic fiber subject to transverse flow in a microfluidic device. Our fiber protrudes into a Hele-Shaw cell, a geometry with a flow field that varies rapidly near the wall but is otherwise approximately uniform. We fabricate our fibers directly in the microchannel using a photopolymerization method. This approach allows us not only to tune the geometry of the fiber (width, length), but also to control the fiber confinement (ratio of fiber height compared to channel height). For varying flow rates we measure how the shape of the fiber changes as a function of its geometry and the confinement. We analyze our results using dimensionless analysis and discuss simplified models of the deflection.

[1]Support from European Project MODIFY.

5:06PM E17.00003 Viscous spread under an elastic lid , JOHN LISTER, JEROME NEUFELD, University of Cambridge, DOMINIC VELLA, University of Oxford — We consider theoretically and experimentally the injection and axisymmetric spread of viscous fluid beneath a flexible elastic lid. In the experiments, glycerol is injected at a constant rate beneath the centre of a 1 cm thick, 50 cm diameter, soft rubber sheet laid on a rigid horizontal surface, which was prewet with an $\approx 200\mu$m thick fluid film. Measurements of the surface elevation and radial propagation are in good agreement with lubrication calculations incorporating bending stresses and gravity. Remarkably, even this simple system evolves through four asymptotic regimes with successive radial spreading laws $r \sim t^{1/6}, t^{7/22}, t^{7/12}$ and $t^{1/2}$. We determine the corresponding prefactors, and confirm the results numerically and experimentally. An alternate problem without the prewetting film is relevant to shallow geological intrusions, called laccoliths, for which we obtain yet more exotic scalings. Our analysis of tip peeling processes in these relatively simple problems gives insight that may find application in more complex problems such as cell adhesion, delamination, and the dynamics of MEMS.

5:19PM E17.00004 Study of mixing at low Reynolds number , R.M. ARCO, R. ZENIT, Universidad Nacional Autonoma de Mexico, E. LAUGA, University of California - San Diego — In many mixing applications is often impossible to operate the mixer in turbulent regime due to the high viscosity of the liquid used (polymers solutions). In this study we propose a technique to induce mixing in laminar flows. A flapper is oscillated at certain frequency and the flow field around it is studied with a PIV technique. The rigidity of the flapper is varied using different materials (acrylic, neoprene). The time-reversibility of the flow was broken when the flapper became flexible. Also we used a Newtonian, and non-Newtonian fluids with elastic effects and nearly constant viscosity. All test were conducted at Re<0.02. Velocity profiles were measured having a impeller oscillation with $\pm$ 60 degrees from the vertical, measurement were taken in the vertical position varying the angular velocities for each impeller. Both the increase of flexibility of the impeller and the increase of the angular velocity was seen to modify the pumping capacity in both fluids. In this talk the nature of this behavior and its implications in mixing processes will be discussed.

5:32PM E17.00005 Flows in Channels with Grooves of Arbitrary Form , ALIREZA MOHAMMADI, JERZY M. FLORYAN, University of Western Ontario — Effects of small-amplitude, two-dimensional grooves on pressure losses associated with flows through channels have been analyzed. The grooves can have an arbitrary inclination with respect to the flow direction. It has been demonstrated that reduced order models of grooves' shapes provides acceptable accuracy. Use of such models permits extraction of the relevant details of geometry and formulation of general conclusions. In most cases replacement of the actual geometry with one Fourier mode of the relevant expansion produced sufficient accuracy. It is shown that the grooves' effects can be divided into those due to the change of the mean position of the wall resulting from addition of the grooves and those due to the flow modulations associated with the shapes of the grooves. The former effects can be determined analytically. The latter effects have been determined numerically using spectrally accurate discretization based on the Fourier and Chebyshev expansions. The results show strong dependence of pressure losses on the groove orientation, with the longitudinal grooves producing the smallest drag and the oblique grooves with the inclination angle of $\sim$42° exhibiting the largest flow turning potential.

5:45PM E17.00006 ABSTRACT WITHDRAWN –

5:58PM E17.00007 A novel release mechanism from responsive microgel capsules[1] , HASSAN MASOUD, ALEXANDER ALEXEEV, George W. Woodruff School of Mechanical Engineering, Georgia Institute of Technology — We use a mesoscale computational model for responsive gels to study the release of nanoparticles from hollow microcapsules. Our model explicitly describes the transport of nanoparticles in swelling/deswelling polymer networks with complex geometries and associated fluid flows. Our simulations show that capsule swelling results in a steady release of encapsulated nanoparticle, which is set by the ability of particles to diffuse through the capsule network. For deswelling capsules, we show that a fluid flow induced by capsule shrinking leads to rapid nanoparticle release. This release, however, is limited due to decreasing mesh size of the deswelling shell. We show that by introducing solid microrods inside deswelling capsules, we can control the rapid release. Our calculations reveal that the rods stretch the deswelling membrane and promote the formation of large pores in the shell, which allow massive flow-driven release of nanoparticles. Thus, our findings reveal a new approach for regulating the release from stimulus responsive micro-carriers that may be useful for designing new drug delivery systems.

[1]Financial support from the Donors of the PetroleumResearchFund, administered by theACS, is gratefully acknowledged.

6:11PM E17.00008 Orthogonal Boundary-Layer Flows , PATRICK WEIDMAN, University of Colorado — A theoretical study is made for boundary-layer flows of different strengths intersecting each other at right angles. Analytic solutions are found for orthogonally interesecting Bickley jets, wall jets, wakes, and uniform shear flows. The equations for intersecting Blasius boundary layers and mixing layers are found and solved numerically. In all cases the development of the boundary-layer thickness for flow in the x-z plane is proportional to a fractional power of (x + z). Extensions of the work are envisioned to include plate transpiration and stretching for the wall-bounded flows.

6:24PM E17.00009 Buckling transition of a flexible fiber at a surface stagnation point , LAURA GUGLIELMINI, Stanford University, NICOLAS AUTRUSSON, Institut Superieur de l'Aeronautique e de l'Espace, AMIT KUSHWAHA, ERIC SHAQFEH, Stanford University, HOWARD STONE, Princeton University — The interplay of viscous and elastic stresses is relevant to a number of flow problems involving slender elastic fibers. These range from the swimming of microorganisms to the transport of pulp fibers in processing flow as well as from nanotube and nanocarpet applications to semi-flexible polymer behavior. In this work, we discuss the response of an elastic fiber tethered to a plane wall and subjected to a stagnation point flow. Using a combination of stability analysis and numerical simulations, (with the latter based on a discretized beam model), we show that, for a critical value of the ratio between viscous and elastic forces, the filament is susceptible to a buckling instability at a bifurcation point which may be subcritical. Further, we discuss the effect of thermal fluctuations on the buckling transition, thus demonstrating the dynamic effect of small fluctuations on a filament whose persistence length is much longer than its contour length.

6:37PM E17.00010 Buckling probability of an elastic fiber transported in a viscous flow of counter-rotating vortices , NAWAL QUENNOUZ, PMMH, YUAN N. YOUNG, New Jersey Institute of Technology, MICHAEL SHELLEY, The Courant Institute, ANKE LINDNER, OLIVIA DU ROURE, PMMH — The interaction of a deformable body with a viscous flow is found in a wide range of situations ranking from biology to polymer science. Here we address the question how an elastic object transported in a viscous flow is deformed by the latter. We experimentally study the deformation and the transport of an isolated elastic fiber in a viscous cellular flow, namely a lattice of counter-rotating vortices, at low Reynolds. We have shown [1] that the fiber can buckle when approaching a stagnation point in this type of flow. The buckling threshold is determined by the relative intensity of viscous and elastic forces, the elasto- viscous number Sp. We observe that even above the threshold the fiber does not buckle each time it passes a stagnation point. We characterize the probability to buckle as a function of the Sp number for a large range of parameters (varying the flow properties, the elastic modulus, aspect ratio of the fiber and length of the fiber compared to the size of the lattice independently). We compare our experimental results to numerical simulations [2] and we discuss the origins of the differences observed. [1] E. Wandersman, N. Quennouz, Fermigier, A. Lindner and O. du Roure. *Buckled in translation*, Soft Matter 6, 57155719, 2010. [2] Y.-N. Young and M. Shelley. *Stretch-Coil Transition and Transport of Fibers in Cellular Flows*, PRL 99, 058303, 2007.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E18 Microfluids: General III 321

4:40PM E18.00001 Tracer Particles Zeta Potential Evaluation from Time Resolved μPIV Measurements , MIQUEL SUREDA, Universitat Politecnica de Catalunya, ANDREW MILLER, FRANCISCO J. DIEZ, Rutgers University — A method is proposed to directly measure the zeta potential of the tracer particles in aqueous solutions using time resolved microPIV (TRμ). The zeta potential of tracer particles in microPIV measurements used in electrokinetic flow analysis in microchannels must be known to accurately calculate the tracer particles electrophoretic velocity. By subtracting this velocity from the observed microPIV velocity, the electroosmotic velocity of the flow is obtained. Using a high-speed laser and camera, the electrophoretic velocity of the tracer particles can be directly calculated from microPIV measurements. The measurements are obtained during the first microsecond of the transient flow that is generated when a potential drop is imposed across a micro-channel. This allows fully visualizing the temporal development of the electroosmotic flow (EOF) in microchannels. For the measurements, 830 nm diameter polystyrene particles are diluted in 100 mM, 10 mM and 1 mM Borate Buffer solutions. A comparison between the proposed TRμ technique and an improved Closed Cell technique validated the results. An advantange of this proposed technique is that measurements can be performed in situ in any microchannel configuration where EOF is needed, without the need for additional calibrations providing reliable data in an easy and quick way.

4:53PM E18.00002 The effect of multivalent counterions on electrokinetically driven flows[1] , NECMETTIN CEVHERI, MINAMI YODA, Georgia Institute of Technology — Various numerical and experimental studies have shown that even trace amounts of multivalent counterions can greatly affect electroosmosis by changing the wall zeta-potential ζ_W via both electrostatic and chemical interactions with the diffuse and Stern layer parts of the electric double layer (EDL). We have previously shown that replacing 1% of the monovalent cation by Ca^{++} can almost halve the electroosmotic mobility, and hence the inferred ζ_W of a symmetric monovalent electrolyte solution [Datta *et al.* (2009)]. These counterions can, however, also affect electrophoresis of the tracers by changing the particle zeta-potential ζ_p. Evanescent-wave particle velocimetry was used to study how trace amounts of divalent cations such as $Ca^{++}$and Mg^{++} affect velocities in the electrokinetically driven flow of various aqueous monovalent electrolyte solutions through channels with a minimum dimension $H \approx 30\mu$m. In all cases, the Debye length $\ll H$. The ζ_p of the tracers were characterized by light scattering, and the steady-state distribution of tracers over the first 300 nm next to the wall was determined from these data to investigate the interactions between the negatively charged particle and wall surfaces and the mobile divalent counterions.

[1]Supported by ARO

5:06PM E18.00003 Interaction of Ion-Concentration Shock Waves in Microfluidics , SUPREET S. BAHGA, ROBERT D. CHAMBERS, JUAN G. SANTIAGO, Stanford University — Electrophoresis based separation techniques, such as capillary electrophoresis and isotachophoresis (ITP), are routinely used in microfluidics to separate ionic species from complex mixtures. Nonlinearities in these electrophoretic processes can result in formation of shock and rarefaction waves. We here focus on shock waves which form in ITP between regions of high and low mobility ions. Depending on the charge of ions, these shocks can propagate either towards anode or cathode, and may interact with each other. We here demonstrate simultaneous anionic and cationic ITP process, in which shock waves approach each other and then interact. Using simulations and experimental visualizations, we show that the interaction of these shock waves can modify the electrophoretic conditions and result in formation of new shock and rarefaction waves. We show two applications where we use shock interaction to couple different electrophoretic processes: (i) where we first preconcentrate DNA fragments in anionic ITP and then use shock interaction to initiate DNA separation, and (ii) where we use shock interaction to elongate ITP zones for higher sensitivity.

5:19PM E18.00004 Concentration polarization and desalination in nanochannels: Effect of surface charge dynamics , MATHIAS B. ANDERSEN, HENRIK BRUUS, Technical University of Denmark, ALI MANI, MARTIN Z. BAZANT, MIT — Mani, Zangle, and Santiago (*Langmuir*, 25, 3898–3916) have shown that at microchannel-nanochannel junctions the coupled effect of concentration polarization and surface conduction can lead to long range propagation of bulk ion-depletion shocks. Essential for this phenomena is the surface charge which for many materials depends on both the concentration and the pH of the local bulk electrolyte. Standard models predict that the surface charge decreases with decreasing concentration leading to the contradictory expectation that there is little or no surface charge in the depleted region and hence no mechanism to sustain long range propagation of desalination shocks. We show that this simple prediction fails to take into account axial transport terms. As such, we couple a surface charge model with the Poisson–Nernst–Planck equations for electric potential and ionic species combined with the Navier–Stokes and continuity equations for fluid velocity. Motivated by experimental work we consider steady-state solutions at the dead end of a nanochannel against a membrane, a scenario where especially space charge and electroosmotic flow are important. Our results suggest that the surface charge density remains finite and does not vanish, and even grows, as the depletion front propagates through the channel.

5:32PM E18.00005 Electromigration dispersion: theory vs. experiment[1] , SANDIP GHOSAL, ZHEN CHEN, Northwestern University — When the concentration of sample ions in zone electrophoresis is significant compared to that of the carrier electrolyte, axial variation of electrical conductivity along the capillary could arise. The consequent variations of the electric field results in electromigration dispersion of the sample peak. It has been shown (S. Ghosal and Z. Chen, Bull. Math. Biol. 2010 vol.72, pg. 2047), that, in an idealized three-ion system consisting ofthe sample ion, a co-ion and a counter-ion, all of equal diffusivity, the time evolution of the sample concentration is governed by a single nonlinear one dimensional transport equation. If the solute concentration is not too large, this equation reduces to Burger's equation that permits exact analytical solution. In the aforementioned analysis, ionic dissociation and recombination was neglected, so that the model only describes systems of strong electrolytes. Here we show that the one dimensional model derived earlier also describes electrophoresis of a solute in the presence of a buffer consisting of a single weak acid (or base). A simple approximate formula is derived for the number of theoretical plates in the presence of electromigration dispersion and compared with published experimental data.

[1]Supported by the NIH under grant R01EB007596.

5:45PM E18.00006 Electromigration dispersion in the presence of a zeta-potential[1] , ZHEN CHEN, SANDIP GHOSAL, Northwestern University — Electromigration dispersion is observed in zone electrophoresis when the concentration of sample ions is comparable to that of the background ions, so that the local electrical conductivity is significantly altered in the sample zone. It was shown (S. Ghosal and Z. Chen Bull. Math. Biol. 2010, vol. 72, pg. 2047) that under certain simplifying assumptions, the concentration profile is described by Burgers' equation. Here we consider a more general situation where the walls of the separation channel may have a non-zero zeta potential and is therefore able to sustain an electro-osmotic bulk flow. The main result is a one dimensional nonlinear advection diffusion equation for the area averaged concentration that accounts for the Taylor-Aris dispersion resulting from the variation in the electro-osmotic slip velocity along the wall. It is shown that in a certain range of parameters, the electro-osmotic flow can actually reduce the total dispersion by delaying the formation of a concentration shock. However, if the electro-osmotic flow is sufficiently high, the total dispersion is increased because of the Taylor-Aris contribution.

[1]Supported by the NIH under grant R01EB007596.

5:58PM E18.00007 Fluid properties in the electrical double layer-effect on streaming potential at charged interfaces , ALEXANDER BARBATI, BRIAN KIRBY, Sibley School of Mechanical and Aerospace Engineering, Cornell University — Fluid properties in the electrical double layer are central to electrokinetic phenomena and affect interfacial measurement. We use a flat plate streaming potential apparatus to measure the interfacial potential of novel (and previously unmeasured) films with unique surface properties. The nature of the parallel plate geometry permits pre- or post-experiment measurement of surface properties (AFM, XPS, etc...) to further interrogate the surface structure. Analytical approximations and a numerical framework are used to directly compute experimental outcomes by combining potential and flow simulations to predict electrical currents generated by the convection of the ion distribution near the liquid-solid boundary; specifically we introduce electric field sensitive variation in the viscosity and permittivity to determine, quantitatively, property effects on linear electrokinetic measurements, with application to novel interfaces and liquids.

6:11PM E18.00008 Velocity field measurements of electrokinetic flow past a conductive cylinder[1] , CETIN CANPOLAT, ALI BESKOK, Old Dominion University — Using the micro particle-image-velocimetry technique, electrokinetic (EK) flow past a conductive circular cylinder (D=0.67 mm) is measured in a rectangular cross-section PDMS/glass microchannel (H=0.1 mm, W=1.0 mm and L=5.3 mm). EK transport in such a system experiences electrophoresis (EP) of the PIV particles, electroosmotic flow (EOF) due to the channel walls, and induced charge electroosmotic (ICEO) flow due to the conductive cylinder. Experiments are conducted using 1xPBS buffer diluted in DI water, and the buffer pH is fixed at 2.05 using HCl solution. This pH value is shown to nearly eliminate the electrophoresis of 0.5 micron carboxylate modified spherical micro-particles used in the PIV studies. Suppression of EP enabled direct measurements of local ICEO flow and its interaction with the global EOF in the channel. By systematically varying the applied electric field from 5 V to 40 V, changes in the velocity field are recorded and correlated with the theoretical trends of EOF and ICEO flow.

[1]C.C. acknowledges the support of TUBITAK for this study.

6:24PM E18.00009 Electrothermal micromixing in 96 well plate , PAUL KAUFFMANN, SOPHIE LOIRE, IGOR MEZIC, UCSB — Diagnostic and pharmacology processes could be greatly accelerated by appropriate mixing. Here electrothermal flows are explored to provide mixing of conductive physiological solutions (=1.6 S/m) in a 96 well plate. Three interdigitated electrodes provide an electric field (< 15Vpp, 1MHz) beneath each well. Polarization and conduction phenomenon of the fluid in a well will be first modeled numerically and compared to an electrical circuit model. Due to high conductivity and permittivity of the fluid, the impedance of the array of filled wells collapse dramatically (96 wells: R = 1Ohm, C=250nF). The power supply challenges accordingly raised by arrays of electrothermal micromixers will be then analyzed. The efficiency of different methods of mixing in those wells will be also compared: the addition of low frequency signal leading to AC electro-osmotic perturbations, a blinking vortices method. The experimental results will be compared to simulations.

6:37PM E18.00010 Electrokinetic flows through a parallel-plate channel with slipping stripes on walls[1]

HENRY C.W. CHU, CHIU-ON NG, Department of Mechanical Engineering, The University of Hong Kong — Electrohydrodynamic flows through a periodically-micropatterned plane channel are considered. One unit of wall pattern consists of a slipping and non-slipping stripe, each with a distinct zeta potential. The problems are solved semi-analytically by eigenfunction expansion and point collocation. In the regime of linear response, the Onsager relation for the fluid and current fluxes are deduced as linear functions of the hydrodynamic and electric forcings. The phenomenological coefficients are explicitly expressed as functions of the channel height, the Debye parameter, the slipping area fraction of the wall, the intrinsic slip length, and the zeta potentials. We generalize the theoretical limits made in previous studies on electrokinetic flow over an inhomogeneously slipping surface. One should be cautious when applying these limits. First, when a surface is not 100% uniformly slipping but has a small fraction of area being covered by no-slip slots, the electroosmotic enhancement can be appreciably reduced. Second, when the electric double layer is only moderately thin, slipping–uncharged regions on a surface will have finite inhibition effect on the electroosmotic flow.

[1]Financial support by the RGC of the HKSAR, China: Project Nos. HKU715609E, HKU715510E; and the HKU under the Seed Funding Programme for Basic Research: Project Code 200911159024.

Sunday, November 20, 2011 4:40PM - 6:24PM –
Session E19 Richtmeyer-Meshkov Instability II 322

4:40PM E19.00001 Simulation of Richtmyer-Meshkov Instability using the Wavelet Adaptive Multiscale Representation (WAMR)

SAMUEL PAOLUCCI, ZACHARY ZIKOSKI, University of Notre Dame — The parallel WAMR method is used to simulate the Richtmyer-Meshkov instability of a shocked and re-shocked thin varicose heavy gas layer. The problem is based on the experiments of Balakumar *et al.*,[1] and modelled by the compressible Navier-Stokes equations for a multi-component gas mixture. The WAMR method provides dynamic, spatial grid adaptivity to efficiently capture features over a wide range of physical scales. Results of the simulation are compared with experimental measurements. Additionally, sensitivity of the long-time behavior of the gas layer to various secondary perturbations is investigated.

[1]Balakumar *et al.* Phys. Fluids **20**, 124103 (2008).

4:53PM E19.00002 The Bipolar Behavior of the Richtmyer–Meshkov Instability[1]

FERNANDO GRINSTEIN, Los Alamos National Laboratory, AKSHAY GOWARDHAN, Lawrence Livermore National Laboratory, RAYMOND RISTORCELLI, Los Alamos National Laboratory — A numerical study of the evolution of the multimode planar Richtmyer-Meshkov instability (RMI) in a light-heavy (air-SF6, Atwood number A=0.67) configuration involving a Mach number Ma=1.5 shock is carried out. Our results demonstrate that the initial material interface morphology controls the evolution characteristics of RMI (for fixed A, Ma), and provide a significant basis to develop metrics for transition to turbulence. Depending on initial rms slope of the interface, RMI evolves into linear or nonlinear regimes, with distinctly different flow features and growth rates, turbulence statistics, and material mixing rates. We have called this the bipolar behavior of RMI.

[1]Funded by LANL LDRD-DR Program on "Turbulence by Design."

5:06PM E19.00003 Theory and simulation of moderately and strongly nonlinear dynamics of the classical Richtmyer-Meshkov instability

M. HERRMANN, Arizona State University, A.L. VELIKOVICH, Plasma Physics Division, NRL, S.I. ABARZHI, The University of Chicago — There are many features of early- and late-time nonlinear Richtmyer-Meshkov instability growth that are not captured by simplified or ad hoc phenomenological models, such as Layzer's or drag-buoyancy. These include but are not limited to: late-time evolution of the bubble curvature, early-time acceleration of the spike, and effect of finite values of ripple amplitude and Atwood number on early-time bubble and spike growth. We compare the results of numerical simulations with the predictions of nonlinear theory, demonstrating good agreement. The influences of viscosity, the initial spectra, and other effects on the nonlinear dynamics are discussed.

5:19PM E19.00004 Numerical Investigation of Turbulence in a Reshocked Richtmyer Meshkov Unstable Curtain of Dense Gas[1]

SANTHOSH SHANKAR, SANJIVA LELE, Stanford University — Moderate resolution numerical simulation of the impulsive acceleration of a dense gas curtain in air by a Mach 1.21 planar shock (modeling the experiments by Balakumar et al. PoF 2008) is carried out by solving the 3-D compressible multi-species Navier-Stokes equation coupled with a localized artificial diffusivity method to capture discontinuities in the flow-field. The simulations account for the presence of three species in the flow-field: air, SF_6 and acetone (used as a tracer species in the experiments). Simulations at different concentration levels of the species are conducted and the temporal evolution of the numerically computed curtain width is compared with the experimental data. The reshock process is studied by re-impacting the evolving curtain with a reflected shock wave. Turbulence statistics computed in the flow-field following reshock are reported and compared with experiment where possible. The reshock time is varied in the simulations to study the turbulence development. Inertial range scaling, vorticity anisotropy and Reynolds stress development are studied in the reshocked flow.

[1]The work is supported by DOE-SciDAC grant DE-FC02-06-ER25787. Computer time was provided by ALCF through the INCITE award.

5:32PM E19.00005 A Computation Study of the RIchtmyer-Meshkov Instability for an Inclined Interface

JACOB MCFARLAND, Texas A&M University, JEFFERY GREENOUGH, Lawrence Livermore National Laboratory, DEVESH RANJAN, Texas A&M University — A computational study of the Richtmyer-Meshkov instability is presented for an inclined interface perturbation in support of experiments to be performed in the Texas A&M shock tube facility. Simulations were performed using the ARES code developed at Lawrence Livermore National Laboratory. A parametric study was performed for two high Atwood number gas pairs, air-SF6 and helium-SF6, where interface inclination angle and incident shock wave Mach number were varied. From this parametric study selected cases were run to late times where reshock occurs. The total circulation for these cases and the circulation production over time is presented. Mix mass and mixing width growth rate are found for before and after reshock.

5:45PM E19.00006 Simulations of Single-mode Richtmyer-Meshkov and Rayleigh-Taylor Instabilities in Spherical Geometries[1] , ANTHONY NELSON, PRAVEEN RAMAPRABHU, University of North Carolina, Charlotte, UNC TEAM — We describe recent numerical simulations of the single-mode Rayleigh-Taylor (RT) and Richtmyer-Meshkov (RM) instabilities in a spherical geometry. The simulations were performed using the astrophysical FLASH code in two- and three-dimensions. Our results are compared to previously published simulation results[2] and linear[3] and nonlinear[4] theories. The RM problem was setup with an air-SF6 interface, subject to a Mach 1.2 shock, and a convergence ratio of 10. We studied the variation of initial amplitude, shock strength, and convergence ratio. We expect the single-mode results to inform multimode growth in applications through bubble merger and competition mechanisms.

[1]The work described here was performed under a grant from the North Carolina Space Grant Consortium.

[2]Sakagami, H. and Nishihara, K., Phys. Rev. Lett. 65, 432, 1990a.

[3]K.O. Mikaelian, Phys. Rev. A., 42 (6), 3400, 1990.

[4]D. S. Clark, M. Tabak, Phys. Fluids 18, 064106, 2006.

5:58PM E19.00007 Turbulent mixing in spherically converging Richtmyer–Meshkov flows , MANUEL LOMBARDINI, D.I. PULLIN, D.I. MEIRON, California Institute of Technology, R.A. GORE, Los Alamos National Laboratory — We discuss large-eddy simulations of the mixing induced at a perturbed, spherical density interface initially impacted by a spherically converging shock wave of Mach number $\simeq 1.2$. Two configurations are investigated: i) air inside and SF_6 (five times denser than air) outside, i.e. heavy–light configuration; ii) SF_6 inside and air outside, or light–heavy configuration. A superposition of spherical harmonics with given power spectrum is used to prescribe an initially small perturbation at the interface while avoiding the pole singularity. Adaptive mesh refinement is employed in the regions of interest around the density interface and shock waves. Interest is focused on the strong turbulent mixing generated by the re-shock. In particular, from data interpolated over spherical surfaces, the power spectra of velocity and density are computed as well as extensive surface-averaged statistics involved in the budget of turbulent kinetic energy and enstrophy density.

6:11PM E19.00008 Reynolds-Averaged Navier–Stokes Model Predictions of Self-Similar Richtmyer–Meshkov Instability-Induced Mixing[1] , OLEG SCHILLING, Lawrence Livermore National Laboratory — A high-order, multicomponent, weighted essentially nonoscillatory implementation of a two-equation K-ϵ Reynolds-averaged Navier–Stokes model is used to simulate reshocked Richtmyer–Meshkov turbulent mixing at various Atwood numbers. The predicted mixing layer evolution is compared with analytical, late-time self-similar solutions of the transport equations. The terms in the transport equation budgets are compared in detail to self-similar profiles across the mixing layer. Additionally, the sensitivity of the turbulence model solutions to variations in the initial conditions and in the model coefficients is explored.

[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.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E20 Instability in Jets and Wakes I 323

4:40PM E20.00001 Competition between single and double helical vortex breakdown , FRANCOIS GALLAIRE, LFMI-EPFL Lausanne, Switzerland, PHILIPPE MELIGA, M2P2-CNRS Marseille, France, JEAN-MARC CHOMAZ, LadHyX, CNRS-Ecole Polytechnique, France — The spiral vortex breakdown of nominally axisymmetric, swirling flows observed in the three-dimensional direct numerical simulations of Ruith, Chen, Meiburg & Maxworthy [JFM 2003, 486, p 331–378] is revisited and interpreted in terms of linearly unstable global modes. We show that increasing the swirl number, which compares the magnitude of the azimuthal and axial velocity components, results in successive Hopf bifurcations involving helical modes of azimuthal wave numbers $m = -1$ and $m = -2$. These modes develop in the lee of the axisymmetric bubble, and correspond to spiral perturbations rotating in time in the same direction as the swirling flow, but winding in space in the opposite direction. The global stability analysis is shown to yield an accurate frequency prediction. We further extend the range of swirl considered in the DNS, and show that the $m = -1$ and $m = -2$ Hopf bifurcations can occur simultaneously, both modes approaching a strong 2:1 resonance. We compute the normal form describing the leading-order nonlinear interaction between modes, and show that it accurately predicts the pattern and symmetries of the solutions observed in the DNS calculations, with satisfactory agreement on the bifurcation thresholds.

4:53PM E20.00002 Experimental study of axial forcing of a swirling jet , AMY MCCLENEY, PHILIPPE BARDET, The George Washington University — Swirling jets enhance mixing of fluids. This leads to more complete combustion, chemical process mixings, lower plume temperatures, and reduced pollutant emissions. Mixing can be improved further by forcing instabilities in the jet and was tested on both non-swirling and swirling jets. These imposed disturbances are either axial, which generates vortex rings or angular, which create more complex structures. In this study, the effects of axial forcing on a swirling jet are experimentally investigated for Reynolds number ranging from 1,000 to 10,000, Swirl number from 0 to 1.3, and a Strouhal number from 0 to 15, where limited past experimental data exists within these flow regimes. This experiment offers insight into the flow structure of the swirling jet in the vortex breakdown region, as observed using PLIF with dye injected azimuthally on the jet periphery.

5:06PM E20.00003 Characterization of hydrodynamic instabilities in non- premixed flames by combining detailed simulations and linear stability analysis , YEE CHEE SEE, MATTHIAS IHME, University of Michigan — Numerical simulations and linear stability analysis are invaluable tools in complementing experiments to obtain improved understanding about instability mechanisms in non-premixed and premixed flames. By utilizing the experiments of Füri et al., the objective of this study is to characterize hydrodynamic instability mechanisms in methane/air diffusion flames. To this end, detailed simulations are performed to obtain the reactive mean flow, which is used as input to the linear stability analysis. In order to account for the detailed reaction chemistry and variations in thermo-viscous properties, a flamelet-representation is introduced in the linear stability method. Model predictions for growth rate and phase velocity are compared with experimental data, and parametric studies are performed to quantify effects of heat-release and detailed chemistry in controlling hydrodynamic instabilities.

5:19PM E20.00004 Dynamics of a laminar flow past a rotating bullet-shaped body[1] , JOSÉ IGNACIO JIMÉNEZ-GONZÁLEZ, ENRIQUE SANMIGUEL-ROJAS, Universidad de Jaen, ALEJANDRO SEVILLA, Universidad Carlos III de Madrid, CARLOS MARTÍNEZ-BAZÁN, Universidad de Jaen — Numerical simulations are performed for laminar flow past a rotating bullet-shaped body of length-to-diameter ratio $L/D = 2$, ranging the non-dimensional angular rotation velocity, $\Omega = \omega R/u_\infty$, from 0 to 0.6, and covering the Reynolds number range $320 < Re < 430$. Three transition patterns stand out as Ω increases on the parametric map Ω- Re. When $\Omega \leq 0.3$, increasing rotation speeds have a stabilizing effect for the first transition, delaying the symmetry breaking bifurcation to higher values of Re, however it triggers the second bifurcation at lower Re than in the case of stationary bodies. Furthermore, the wake undergoes three different flow regimes as Re increases: axisymmetric, *frozen* (i.e. two counterrotating vortices aligned with the streamwise direction rotate around the axis with a different frequency than that of the body) and spiral flow. A different behavior has been observed in the range $0.3 \leq \Omega \leq 0.5$ where only a single transition exists. Finally, when $\Omega > 0.5$ there are also two transitions, both leading to different swirling flows.

[1]Supported by the projects DPI2008-06624-C02, P07-TEP02693 and P11-TEP5702.

5:32PM E20.00005 Oblique and Parallel Vortex Shedding in the Wake of a Circular Cylinder at Subcritical Flow Regime , ANTRIX JOSHI, TAYFUN AYDIN, ERIC KHOURY, ARMINA ANDRIASIAN, ALIS EKMEKCI — Parallel/oblique shedding of vortices in the wake of a cylinder at subcritical Reynolds numbers are investigated experimentally in a water channel. Experiments involved a cylinder with various combinations of free-surface, channel-floor, endplate boundaries, such as, a free-surface on one end and the channel floor on the other end of the cylinder. The cylinder aspect ratios (L/D) ranged between 12.3 to 13.5. Flow visualization via PIV is conducted along the spanwise flow field on a plane that is tangent to the cylinder surface and is oriented in the free-stream direction. As the vortices are shed behind this plane of visualization from sides of the cylinder, distinct spanwise regions with increased or decreased levels of streamwise velocity contours are alternately induced. These regions can be used to estimate the spanwise orientation of the vortices. Our preliminary results showed an unsteady character in the degree of obliqueness of the shed vortices. Vortices shed out of phase between two neighboring regions of the span led to the formation of dislocations.

5:45PM E20.00006 Instabilities of a cylinder wake in a stratified fluid , PATRICE MEUNIER, IRPHE, Aix-Marseille Univ,, CNRS — The goal of this study is to analyse experimentally, numerically and theoretically how a linear density stratification modifies a cylinder wake, which is well known to exhibit a rich dynamics in a homogeneous fluid. In a first part, we focus on the 2D dynamics of the wake. We show that the von Karman vortex street is stabilised by a moderate stratification for tilted and horizontal cylinders, in agreement with the stabilisation of shear flows at large Richardson numbers. However, it is curious to see that the von Karman vortices reappear for a strong stratification in the case of a tilted cylinder. This new unstable mode can be explained by the presence of tilted vortices with no vertical velocity, i.e. with horizontal streamlines. In a second part, we focus on the 3D instabilities of the cylinder wake. For a vertical cylinder, the well known mode A can be nicely visualised by shadowgraph and seems to be enhanced by a moderate stratification. For a tilted cylinder, the structure of the instability is strongly modified, with the presence of thin undulated dark lines in the shadowgraph images. These structures are similar to the Kelvin-Helmholtz billows which have been observed recently in the critical layer of a tilted stratified vortex.

5:58PM E20.00007 Sensitivity Analysis of Spiral Vortex Breakdown , UBAID QADRI, DHIREN MISTRY, MATTHEW JUNIPER, University of Cambridge — Vortex breakdown occurs in some swirling flows, such as those in gas turbine combustion chambers. Previous studies have established that the initial breakdown is steady and axisymmetric but that an unsteady spiralling breakdown mode develops on top of this, due to a region of absolute instability. We investigate the linear stability of steady axisymmetric vortex breakdown in a semi-infinite domain for an incompressible fluid at Re = 200. We relate the global behaviour of the flow to its local stability properties. We use direct numerical simulation (DNS) of the linearized direct and adjoint Navier–Stokes equations to obtain the linear direct and adjoint global modes. We use these to map the regions of the flow that are most sensitive to external forcing and internal feedback. This enables us to identify the wavemaker region, which causes spiral vortex breakdown. We find that, for low swirls, the wavemaker of the linear global mode lies in the axisymmetric breakdown bubble. Previous studies of the same flow indicate that the wavemaker of the nonlinear global mode lies in the wake of the axisymmetric breakdown bubble. We explain this apparent contradiction by analogy with two coupled Van der Pol oscillators.

6:11PM E20.00008 Stability and sensitivity analysis of experimental flow fields measured past a porous cylinder , SIMONE CAMARRI, Dept. of Aerospace Engineering - University of Pisa, JENS H.M. FRANSSON, BENGT E.G. FALLENIUS, Linné Flow Centre, KTH Mechanics — It is known in the literature that the linear stability analysis of the time-averaged flow field past a circular cylinder, after the primary wake instability, predicts a global mode that is marginally stable with a frequency in time that well approximates the one of the saturated vortex shedding. This behavior has been recently shown to hold up to Reynolds number $Re = 600$ by DNS simulations [Leontini et al., JFM 645, pp. 435-446, 2010]. Here we carry out a stability/sensitivity analysis of the PIV velocity fields measured in the wake past a porous circular cylinder at $Re \simeq 8.3 \cdot 10^4$ [Fransson et al., JFS 19, pp.1031-1048, 2004]. Different intensities of uniform suction/blowing through the cylinder surface are considered. The objectives of this work are the following. Firstly, we extend the analysis described in [Leontini et al.] at higher values of Re. Moreover, the global direct and adjoint modes, derived from the experimental data, are used to locate the core of the instability, to realign the instantaneous flow snapshots in phase and, thus, to help in the analysis of the experimental data. Lastly, it is shown that the sensitivity of the marginally stable eigenvalue to a generic variation of the mean flow provides hints for the control of the vortex-shedding frequency.

6:24PM E20.00009 Flow Structure and Stability in Confined, Reacting, Bluff Body Wakes , BENJAMIN EMERSON, Georgia Institute of Technology — This paper describes the variation of reacting bluff body wake structure with flame density ratio for a variety of bluff bodies and lip velocities. Previous experiments and computations have shown that the bluff body flow structures at "high" and "low" flame density ratios are fundamentally different, being dominated by the convectively unstable shear layers and absolutely unstable Von Karman vortex street, respectively. This paper characterizes the transition between these two flow structures, and shows that the bifurcation behavior does not occur abruptly at some density ratio. Rather, there exists a range of transitional density ratios at which the flow exists intermittently in both states. The fraction of time that the flow spends in either state is a monotonic function of density ratio. This paper also shows that local parallel stability analyses developed for confined, variable density, laminar base wake flows captures the qualitative behavior of this transition, demonstrating a successful parameterization of this phenomenon. These results have important implications on the dynamics of high Reynolds number, vitiated, reacting flows, suggesting that such flows exhibit two co-existing dynamical states, intermittently jumping between the two.

6:37PM E20.00010 Measurements in a bluff body wake with variable inlet condition[1] , BENGT E.G. FALLENIUS, RENZO TRIP, JENS H.M. FRANSSON, Linne Flow Centre, KTH Mechanics — The aim of this project is to experimentally study the instability of wakes behind bluff bodies from a fundamental research point of view, both for the natural case as well with various flow control methods applied. This is realized in an experimental setup specially designed to perform parameter variations, which are most often not possible in usually fixed experimental geometries. The bluff body is a so-called rectangular-based forebody with permeable surfaces on both sides, which enables modulation of the boundary layer through suction or blowing of air through the surfaces. The suction or blowing can either be uniform or local and the two sides can be modulated independently and consequently the inlet condition of the wake is altered. Investigations have been performed by means of pressure measurements, hot-wire anemometry and PIV in order to study how the wake mean flow, the vortex shedding frequency and the vortical structures are affected by the wake initial condition.

[1]This work was financially supported by the Swedish Research Council and the Göran Gustavsson Foundation. Travel grant by the Bengt Ingeström Foundation is also acknowledged.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E21 Particle-laden Flows II 324-325

4:40PM E21.00001 Inertial particle clustering in local linear flow: Novel satellite particle simulations

, BAIDURJA RAY, LANCE COLLINS, Cornell University — At spatial scales below the Kolmogorov length-scale, a turbulent flow around an inertial particle can be adequately approximated by a locally linear flow field, with the velocity at any point being determined solely by the velocity gradient at the particle location. Here, we describe novel satellite particle simulations, where inertial particle clustering is studied at length-scales below the Kolmogorov scale (η). Such simulations allow us to isolate the effect of the sub-Kolmogorov scales on particle statistics. We show that such simulations capture the correct qualitative behavior of the radial distribution function (RDF, a measure of particle clustering) with Stokes number (St) and separation distance (r/η) and accurately predict the power (c_1) in the power-law behavior of the RDF ($g(r/\eta) = c_0(r/\eta)^{-c_1}$) for $St \leq 0.4$. We also test a drift-diffusion model for particle motion and show that it correctly predicts the power-law for $St \leq 0.4$. We also investigate the effect of velocity filtering and show that the satellite simulations are able to capture the effect of filtering on the slope c_1 of the RDF. Our results provide insight on the effect of small-scales on inertial particle statistics and guide us towards modeling the role of sub-Kolmogorov scales on particle motion.

4:53PM E21.00002 Computational and experimental analysis of particle clustering in a shearless turbulent mixing layer

, PETER IRELAND, GARRETT GOOD, ZELLMAN WARHAFT, LANCE COLLINS, Cornell University — Entrainment, the drawing in of external fluid by a turbulent flow, is ubiquitous to both industrial and natural turbulent processes. This mechanism is particularly important in atmospheric clouds, where the entrainment of dry air by turbulence can affect precipitation mechanisms. We use parametrically matched wind-tunnel experiments and direct numerical simulations with inhomogeneous particle seeding to explore particle clustering in a shearless turbulent mixing layer. We find high degrees of clustering, both visually and statistically, even for particles with negligible inertia. These particle clusters have characteristic sizes on the order of the integral lengthscale of the turbulence and are thus much larger than those resulting from particle inertia. The degree of clustering at a particular location generally decreases as the mixing layer evolves and depends on both the turbulent kinetic energy ratio in the mixing layer and the magnitude and orientation of gravity. We observe the same qualitative trends in both the experiments and the simulations. We anticipate that a better understanding of particle clustering in entraining flows will lead to, among other things, improved cloud evolution predictions and more accurate climate models.

5:06PM E21.00003 Computational and experimental study of charged inertial particles in turbulence

, JIANG LU, Physics Department, Michigan Technological University, HANSEN NORDSIEK, Physics Department , University of Maryland, RAYMOND SHAW, Physics Department ,Michigan Technological University — We have investigated the behavior of electrically charged, inertial particles in homogeneous, isotropic turbulence. Specifically, a drift-diffusion theory describing the scale dependent radial distribution function (RDF) for the particles is evaluated computationally and experimentally. The experiments were carried out in a laboratory chamber that generates nearly homogeneous, isotropic turbulence, and seeded with uniformly charged water droplets. We also report results from a direct numerical simulation (DNS) of turbulence in a periodic box using the pseudospectral numerical method. The numerical study explicitly calculates 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, sufficiently strong so as to mimic behavior of a nonideal gas: Coulomb interactions lead to short range repulsion overcoming inertial clustering below a shielding length as seen by a strong reduction in the RDF, but turbulence is sufficiently intense so as to suppress long range correlations (e.g., analogous to thermal energy leading to melting of a Coulomb crystal).

5:19PM E21.00004 Experimental and Numerical investigation of a droplet-laden turbulent flow: preferential concentration due to turbulence and its influence on droplet collisions and growth

, ALBERTO ALISEDA, COLIN BATESON, University of Washington, ORLANDO AYALA, HOSSEIN PARISHANI, LIAN-PING WANG, University of Delaware, BOGDAN ROSA, IMWM, Poland — We have conducted a multi-laboratory investigation of the dynamics of small inertial droplets ($St \approx 0.1 - 1$) immersed in homogeneous isotropic turbulence. We compare experimental results from a wind tunnel experiment with Direct Numerical Simulations of slowly decaying homogeneous isotropic turbulence laden with spherical droplets. The Reynolds number (Re_λ) in both cases is of the order of 200 and the particle distribution is polydisperse with droplets in the $10 - 30$ μm range. We compare the one-dimensional Radial Distribution Function from the experiments to the 1D, 2D and 3D RDFs from the simulations to validate the numerical treatment of the droplet dynamics in close proximity, and to develop methods to extrapolate the experimental measurements to 3D. We also compare the relative velocity of a pair of droplets, obtained along a line or in a plane from PDPA and PIV measurements, to the equivalent statistics obtained from the 3D velocity fields in the DNS. These are key components of the droplet collision kernel necessary to calculate turbulence-induced collision-coalescence and droplet growth.

5:32PM E21.00005 Highly scalable parallel implementation of turbulent collision of aerodynamically interacting cloud droplets[1]

, HOSSEIN PARISHANI, ORLANDO AYALA, LIAN-PING WANG, University of Delaware, BOGDAN ROSA, Institute of Meteorology and Water Management, Poland, WOJCIECH GRABOWSKI, NCAR — Hybrid direct numerical simulation (HDNS) has advanced our understanding of turbulent collision-coalescence of cloud droplets. In this approach, the background fluid turbulence is simulated by a pseudospectral method and disturbance flows of droplets are treated analytically. To better realize its potential on PetaScale computers with ∼100,000 processors, here we implement and test a parallel implementation using two-dimensional domain decomposition. The purpose is to increase both the range of flow scales and the number of droplets realizable in the simulations, so the dependence of collision statistics on flow Reynolds number and droplet size can be explored. We expect that the 2D domain-decomposition HDNS code can be used to produce statistics of aerodynamically-interacting droplets with Taylor microscale flow Reynolds number R_λ up to ~ 1000 and a system of $\mathcal{O}(10^7)$ polydisperse droplets. We will present the implementation details as well as results of turbulent collision statistics (e.g., collision kernel, radial distribution function, relative velocity statistics) of sedimenting cloud droplets from our latest high-resolution HDNS.

[1]Work supported by NSF and NCAR.

5:45PM E21.00006 The effect of dissipation intermittency on the turbulent collision statistics of cloud droplets

, LIAN-PING WANG, University of Delaware, WOJCIECH GRABOWSKI, NCAR — Atmospheric cloud turbulence has a Taylor microscale flow Reynolds number typically in range of 10^4 to 10^5, therefore, the local fluid acceleration and local dissipation rate are highly intermittent. The collision rate of cloud droplets and related pair statistics (i.e., the radial relative velocity and radial distribution function) are affected by these local flow intermittency, therefore, it has been speculated that the flow intermittency might have a significant impact on the turbulent collision of cloud droplets. In this talk, we argue, however, that this speculation may not hold for several reasons. First, collision is a binary interaction and its average statistics are governed by second-order statistical moments, the average impact of intermittency is not significant for these lower order statistics. Second, while the intermittent regions of high flow acceleration and high local dissipation rates do occur, they occupy a very insignificant volume with short lift time. Third, for given droplet sizes, the range of scales governing the pair statistics is limited. An analysis is developed to assess the effect of flow intermittency on the average collision kernel, showing that the flow intermittency does not have a significant effect. Other related works and results from direct numerical simulations will also be used to support this finding.

5:58PM E21.00007 A Lagrangian clustering analysis of inertial particles in turbulence using three-dimensional Voronoï tessellations , YOSHIYUKI TAGAWA, JULIAN MARTINEZ MERCADO, VIVEK N. PRAKASH, The University of Twente, ENRICO CALZAVARINI, Universit é Lille, CHAO SUN, DETLEF LOHSE, The University of Twente — Three-dimensional Voronoï analysis is used to quantify the clustering of inertial particles in homogeneous isotropic turbulence using data sets from numerics in the point particle limit and one experimental data set. We study the clustering behavior at different density ratios, particle response times (i.e. Stokes numbers St) and several Taylor-Reynolds numbers. The Probability Density Functions (PDFs) of the Voronoï cell volumes of light and heavy particles show a different behavior from that of randomly distributed particles, implying that clustering is present. The results are consistent with previous investigations. The small Voronoï volumes of light particles correspond to regions of higher enstrophy than those of heavy particles, indicating that light particles cluster in higher vorticity regions. The Lagrangian temporal autocorrelation function of Voronoï volumes shows that the clustering of light particles lasts much longer than that of heavy or neutrally buoyant particles. Due to inertial effects arising from the density contrast with the surrounding liquid, light and heavy particles remain clustered for much longer times than the flow structures which cause the clustering.

6:11PM E21.00008 Inertial particles in turbulent convection , VALENTINA LAVEZZO, HERMAN J.H. CLERCX, FEDERICO TOSCHI, Eindhoven University of Technology, Eindhoven, The Netherlands — Particle re-suspension mechanism in a non-isothermal flow has direct relevance for many industrial and environmental applications, where the properties of the flow (heat transfer, turbulence, etc.) can be modified by the presence of particles. A Lattice Boltzmann Method coupled with Lagrangian particle tracking is used to investigate the behaviour of inertial particles released in a horizontally periodic turbulent Rayleigh-Bénard convection cell. In particular, we focus on the effects of plume formation and turbulent structures on particle re-suspension in the domain. Different Froude numbers are considered to evaluate the influence of inertia and gravity on particle entrainment and dispersion. A two-way coupling approach has been included to estimate the impact of the presence of particles on the heat exchange between the two walls. Statistics revealed a close relationship between particle behaviour and fluid thermal structures and therefore a strong variation of the temperature field, when a back reaction of the particles onto the fluid is considered. Mean and higher order statistics on particle and fluid velocity and temperature fields are also presented.

6:24PM E21.00009 Effect of gravity on the dispersion of heavy particles[1] , YONGNAM PARK, YONGRAK JUNG, CHANGHOON LEE, Yonsei University — The effect of gravity on the dispersion of heavy particles is investigated in direct numerical simulation of forced isotropic turbulence at moderate Reynolds numbers. The integral time scales of fluid seen by particle show different behavior from that of Lagrangian fluid for wide range of Stokes number and magnitude of gravity. Autocorrelation functions of velocity of heavy particles become reduced when stronger gravity is applied, implying the reduction of dispersion. Integral time scales of fluid seen by particle also get smaller when gravity is stronger. Compared to the direction of gravity, normal direction velocities show shorter correlation. Settling velocity of heavy particle gets smaller when the Stokes number goes to zero or gravity gets smaller. Gravity enhances the preferential concentration of heavy particle and different type of large-scale nonuniform distribution of particles is clearly observed for the Stokes number of 10 when strong gravity is exerted. The preferential concentrations by strong gravity are not related with the turbulence structure and relevant physical explanation will be discussed in the meeting.

[1]Authors acknowledge support from ERC program sponsored by National Research Foundation of Korea.

6:37PM E21.00010 Anisotropy in Pair Dispersion of Inertial Particles in Turbulent Channel Flow , ENRICO PITTON, CRISTIAN MARCHIOLI, ALFREDO SOLDATI, University of Udine, VALENTINA LAVEZZO, FEDERICO TOSCHI, Technische Universiteit Eindhoven — The rate at which two particles separate in turbulent bounded flows is crucial to predict the inhomogeneities of particle spatial distribution and to characterize mixing. In this paper we examine the role of mean shear and small-scale turbulent velocity fluctuations on pair separation. To this aim an Eulerian-Lagrangian approach based on pseudo-spectral DNS of turbulent channel flow is used. Pair separation statistics were computed for particles with different inertia (and for tracers) released from different regions of the channel. Results confirm that shear-induced effects predominate when the pair separation distance becomes comparable to the largest scale of the flow, and also reveal the fundamental role played by particles-turbulence interaction at the small scales in triggering separation during the initial stages of pair dispersion. As a result, pair dispersion in non-homogeneus anisotropic turbulence has a superdiffusive nature and may generate non-Gaussian number density distributions of both particles and tracers. These features persist even when statistics are shown free of unidirectional convection, and exhibit strong dependency on particle inertia.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E23 Reacting Flows II: Detonations & Supersonic Combustion 326

4:40PM E23.00001 Hydrogen-air detonation cells computed using skeletal and reduced reaction mechanisms , DAVID KESSLER, BRIAN TAYLOR[1], VADIM GAMEZO, ELAINE ORAN, Naval Research Laboratory — The multidimensional instability of gas-phase detonations results in a complex dynamic structure at the detonation front that leaves behind characteristic cellular patterns as it propagates. In fuel-air mixtures with high effective activation energies, such as hydrogen and air, these detonation cells can become irregular and modelling this behavior using reduced chemical reaction mechanisms can be challenging. Using complex reaction mechanisms, however, can be computationally overwhelming for problems of practical interest. We compare the reaction front structures and dynamic behavior of two-dimensional detonations in a stoichiometric hydrogen-air mixture computed using a 12-step skeletal mechanism and several reduced mechanisms, including a calibrated one-step model. We pay particular attention to how transverse instabilities that form in this high-activation-energy mixture are affected by the details of the chemistry model. We then discuss how to adjust the parameters in reduced reaction models to better describe irregular triple point behavior.

[1]NAS/NRC Postdoctoral Research Associate

4:53PM E23.00002 Numerical Simulations of Detonation Instabilities and Magnetic Field Interactions[1] , LORD COLE, HAI LE, ANN KARAGOZIAN, UCLA — Numerical simulations of magneto-hydrodynamic (MHD) effects on high frequency and low frequency one-dimensional detonation wave instabilities are performed, with applications to flow control and MHD thrust augmentation in Pulse Detonation Engines and their design variations.[2] The dynamics of the hydrogen-air detonation are explored via high order shock capturing schemes and complex reaction kinetics. The flame is initially strongly coupled to the shock and the wave is over-driven. As the degree of overdrive decays and the detonation approaches the CJ limit, high frequency instabilities begin to appear. Eventually the average induction length continues to increase and a second mode can be seen which directly couples the flame speed with the shock, resulting in fluctuations with lower frequency but much higher amplitude. A simple model for flame-shock coupling replicates the quantitative features of these instabilities quite well. Effects of an externally applied magnetic field on these detonation instabilities are explored. In addition, the complex chemical kinetics calculations are ported onto a GPU, and computational performance may be compared with standard CPU-based computations.

[1]Supported by AFOSR and AFRL/RZSS.

[2]Zeineh, et al., **J. Propusion & Power**, to appear

5:06PM E23.00003 Structure of detonation waves in mixtures containing chemical inhibitors

, MILTIADIS V. PAPALEXANDRIS, Universite Catholique de Louvain — In this talk we present numerical results of detonation transmission in mixtures containing chemical inhibitors. In our study, the chemical kinetics model consists of a three-step chain-branching mechanism for the fuel combustion and an one-step mechanism for the endothermic reaction between inhibitor and radical. Results from both one and two-dimensional simulations are discussed. It is shown that the absorption of heat and the consumption of radical in the endothermic step result in longer induction zones. This leads to a temporary detachment of the reaction zone from the precursor shock. Eventually, the material behind the shock starts to burn rapidly, thus producing pressure waves which reach the precursor shock and re-initialize the detonation. This is followed by large over-pressures and highly irregular oscillations of the shock. The talk concludes with a discussion on the necessary inhibitor concentration in the initial mixture to achieve permanent detonation suppression.

5:19PM E23.00004 Deflagration-to-Detonation Transition in Unconfined Media

, ALEXEI POLUDNENKO, Naval Research Laboratory, THOMAS GARDINER, Sandia National Laboratories, ELAINE ORAN, Naval Research Laboratory — Deflagration-to-detonation transition (DDT) can occur in environments ranging from experimental and industrial systems on Earth to astrophysical thermonuclear supernovae explosions. In recent years, substantial progress has been made in elucidating the nature of this process in confined systems with walls, obstacles, etc. It remains unclear, however, whether a subsonic turbulent flame in an unconfined environment can undergo a DDT. We present simulations of premixed flames in stoichiometric H_2-air and CH_4-air mixtures interacting with high-intensity turbulence. These calculations demonstrate the DDT in unconfined systems unassisted by shocks or obstacles. We discuss the mechanism of this process and its implications.

5:32PM E23.00005 Richtmyer-Meshkov instability in shock-flame interactions

, LUCA MASSA, University of Texas at Arlington, PALLAV JHA COLLABORATION — Shock–flame interactions occur in supersonic mixing and detonation formation. Therefore, their analysis is important to explosion safety, internal combustion engine performance, and supersonic combustor design. The fundamental process at the basis of the interaction is the Richtmyer-Meshkov instability supported by the density difference between burnt and fresh mixtures. In the present study we analyze the effect of reactivity on the Richtmyer- Meshkov instability with particular emphasis on combustion lengths that typify the scaling between perturbation growth and induction. The results of the present linear analysis study show that reactivity changes the perturbation growth rate by developing a non-zero pressure gradient at the flame surface. The baroclinic torque based on the density gradient across the flame acts to slow down the instability growth for high wave numbers. A non-hydrodynamic flame representation leads to the definition of an additional scaling Peclet number, the effects of which are investigated. It is found that an increased flame-contact separation destabilizes the contact discontinuity by augmenting the tangential shear.

5:45PM E23.00006 Thermomechanics in an Externally Heated Reactive Gas Hot spot

, DAVID R. KASSOY, University of Colorado, Boulder — The response of a finite volume of inert gas to spatially resolved, transient power addition on a time scale short compared to the local acoustic time has been described by Kassoy, J. Eng. Math., **68**, 249-262. Formal parameter asymptotic methods are employed to describe the thermodynamic changes caused by a relatively brief burst of thermal energy, as well as the induced mechanical response. Related methods are used in the present work to describe similar physical processes occurring in a heated reactive gas capable of a high activation energy reaction. Localized power deposition from an external source to a spherical volume of reactive gas causes **nearly** constant volume heat addition. The spatially distributed temperature and pressure increase together, accompanied by an induced low Mach number gas expansion and a concomitant small decrease in fluid density. The resulting low Mach number velocity of gas expelled from the sphere surface is the source of acoustic disturbances, confined to a thin shell surrounding the finite volume, on the short heating time scale. A spatially distributed thermal explosion is initiated on a much longer induction time scale within the hot spot. Thermodynamic and gas dynamic consequences of the subsequent chemical heat addition are identified. Results are compared with those of Vasquez-Esp' and Linan (CTM,**5**, 485-498) found from a more intuitive formulation using coordinate expansion asymptotic methods.

5:58PM E23.00007 Direct numerical simulation of supersonic combustion with finite-rate chemistry[1]

, AMIRREZA SAGHAFIAN, HEINZ PITSCH, Stanford University — Three-dimensional direct numerical simulations (DNS) of reacting and inert compressible turbulent mixing layers have been performed. The simulations cover convective Mach numbers from subsonic to supersonic. A detailed chemistry mechanism with 9 species and 29 reactions for hydrogen is used in the reacting simulations. Effects of different initial conditions on the structure of the mixing layer, and time required to reach self-similarity are studied. Flame/turbulence interaction is analyzed by studying turbulent kinetic energy, Reynolds stresses, and their budgets in the reacting and inert simulations. The effects of different reactions on the heat release and mixture composition especially in the regions where shocklets impinge the flame are studied. These DNS databases will provide a better understanding for the compressibility effects on the combustion, and will be used to assess the accuracy of Flamelet/Progress variable approach in supersonic regime.

[1]This material is based upon work supported by the Department of Energy under the Predictive Science Academic Alliance Program (PSAAP) at Stanford University, Award Number(s)DE-FC52-08NA28614.

6:11PM E23.00008 Including compressibility effect in combustion modeling for LES of supersonic combustion

, RONAN VICQUELIN, Ecole Centrale Paris, JOHAN LARSSON, JULIEN BODART, Stanford U. — Predicting scramjet performance requires accurate estimations of heat losses, wall friction and heat release distribution. The combustion modeling strategy is therefore important although most numerical simulations of these configurations use simple approximations to model the reactive flow. Two tabulated chemistry models are used here to perform LES of a model scramjet combustor. A wall model is used to compute heat losses and friction at walls. The first combustion model is based on non-premixed flamelets with a progress-variable approach. The compressibility effects of the flow on chemistry are taken into account by rescaling the progress-variable source term with ad-hoc laws. The second model extends the first chemical table by adding pressure and enthalpy dimensions, which enables to give a better estimation of the compressibility effect on the flame. LES results are compared to experimental data (OH LIF, OH* and pressure measurements) to assess the importance of dealing with compressibility effects in combustion models.

6:24PM E23.00009 Flame structure in a compact inlet/scramjet combustor[1]

, MIRKO GAMBA, VICTOR A. MILLER, M. GODFREY MUNGAL, RONALD K. HANSON, Stanford University — OH PLIF is used to investigate the flame structure in an inlet/scramjet combustor model tested in the Stanford 6-Inch expansion tube. The model is characterized by a compression inlet that generates a shock train confined within the constant area combustor. The inlet flow is maintained at $M = 2.8$, $p = 40kPa$ and $T \sim 1250K$. Hydrogen is injected through a single transverse injector at equivalence ratios up to 0.44. Optical access along the combustor allows flow imaging on orthogonal planes. Global effects of fuel injection are quantified by wall-pressure measurements with oxidizing and non-oxidizing freestreams. It is found that mass addition alone does not significantly impact the flowfield. Fuel injection and combustion effects manifest well downstream even under conditions where combustion is stabilized at the injector. The general flame structure in the near-field of the jet shares many similarities with isolated transverse jets. Similarly, in spite of the shock train, the flame structure in the wake region behaves as an undisturbed shear layer where a laminar-like structure typical of low-speed nonpremixed combustion is observed. Finally, the results suggest that the shock train may favor flame stability and anchoring at the shock reflection points.

[1]Sponsored by DoE PSAAP at Stanford University.

6:37PM E23.00010 LES of supersonic combustion in a realistic scramjet combustor , JOHAN LARSSON, RONAN VICQUELIN, IVAN BERMEJO-MORENO, JULIEN BODART, Stanford University — Large eddy simulation is used to study the reacting flow in the HyShot scramjet combustor. The H2/air chemistry is modeled using a flamelet/progress-variable approach, which is adapted to the supersonic regime through a parametrized rescaling with pressure of the progress-variable source term. An algebraic wall-model is used to alleviate the need to resolve the inner part of the boundary layers, while still capturing the large momentum and energy losses to the cooled walls. Results are shown for two geometries and compared to the limited experimental data available for each: a model combustor with OH LIF, OH* and wall pressure measurements, as well as the HyShot scramjet combustor with mainly wall pressure data.

Sunday, November 20, 2011 4:40PM - 6:50PM –
Session E26 Rotating Flow II 329

4:40PM E26.00001 Spatial variation of angular momentum transport in turbulent spherical Couette flow.[1] , DANIEL ZIMMERMAN, SANTIAGO TRIANA, DANIEL LATHROP, University of Maryland — Turbulent shear flow between two concentric rotating spheres of radius ratio $\eta = 0.35$ is studied up to 56 million Reynolds number. The dimensionless differential rotation as expressed by the Rossby number $Ro = (\Omega_i - \Omega_o)/\Omega_o$ is an important parameter in setting the observed large scale flow state and the angular momentum flux through the system at a given Reynolds number $Re = (\Omega_i - \Omega_o)L^2/\nu$. Here we consider the case where the inner sphere super-rotates in the range $0 < Ro < 62$. Using torque measurements on the inner sphere and local wall shear stress measurements on the outer sphere, we infer a high latitude concentration of azimuthal wall shear stress peaking around $Ro = 7$ that coincides with a minimum in the dimensionless torque relative to that for the flow at $Ro = \infty$ (outer sphere at rest). Intermittent transitions between different states of large scale flow and system-scale waves show similar anti- correlation between the local shear stress on the outer sphere and the torque on the inner sphere, and we discuss the interpretation of these dynamics and possible implications for other rotating shear flows, like Taylor-Couette flow.

[1]Supported by the National Science Foundation.

4:53PM E26.00002 Experimental Quantification of Inverse Energy Cascade in Deep Rotating Fluid , EHUD YAROM, ERAN SHARON, The Hebrew University of Jerusalem — We present quantitative experimental measurements of an inverse energy cascade in deep rotating turbulence. Experiments were performed in a rotating cylindrical water tank configuration with energy injection at a large wavenumber, k_i. For $k<k_i$ the steady state energy spectrum is very well described by $E_k = C\varepsilon^{2/3}k^{-5/3}$, the 2D energy spectrum. The temporal evolution of energy spectra towards steady states was measured. We identify a "front" which propagates from large to small wave numbers, behind which the spectrum attains its steady state value. The propagation of these fronts was measured for different energy injection rates and rotation frequencies. These measurements are in good quantitative agreement with the predictions for 2D turbulence.

5:06PM E26.00003 Taylor-Columns in Deep Water: Testing the Limits of the Taylor-Proudman Theorem , JOZEF H.A. VLASKAMP, PETER J. THOMAS, University of Warwick, RAINER HOLLERBACH, University of Leeds, ROBERT M. KERR, University of Warwick — The formation of Taylor-Columns is one of the familiar phenomena observed in flows where strong background rotation is present. The current investigation considers the Taylor-Column generated by a differentially rotating disk in a rotating fluid, a geometry similar to the classic Stewartson-layer problem. Experimental work was performed on the large turntable at the University of Warwick (overall height 5.7m and 1.4m diameter). It offers a water depth of 2m below the disk, allowing for a much longer Taylor-Column to be observed than in previous experimental studies. An automated, traversing Stereo-PIV system was developed to visualize the flow, allowing data-acquisition at different heights in the tank. The experimental results show a z-dependence of the angular velocity in the interior of the Taylor-Column for positive differential rotation, but not for negative differential rotation. These results contradict both the Taylor-Proudman theorem and numerical predictions.

5:19PM E26.00004 Experimental modelling of bistability in mid-latitude atmospheric jets[1] , JOEL SOMMERIA, MANIKANDAN MATHUR, Laboratoire des Ecoulements Geophysiques et Industriels, Grenoble, France, FREDDY BOUCHET, Laboratoire de Physique, ENS de Lyon, Lyon, France — Bistability, a striking property of some climate features, can be obtained in purely hydrodynamic systems in turbulent regimes and can be explained in terms of statistical mechanics. Here, we report laboratory experiments reproducing mid-latitude atmospheric jets in an annular rotating tank. The jet is influenced by a non-axisymmetric bottom topography interfering with the natural wavy pattern of the jet, which is related to a barotropic instability. Two states emerge from this interaction: a wave 'blocked' with respect to the topography, and a freely propagating wave. These two states are obtained with the same control parameters, the chosen state depending on the history of the system. No spontaneous transition between the two states is observed, in spite of the strong turbulent fluctuations. Finally, the response of the system to external noise will be presented.

[1]Funding from the project ANR STATOCEAN is gratefully acknowledged.

5:32PM E26.00005 Vortex formation through inertial wave focusing[1] , MATIAS DURAN-MATUTE, JAN-BERT FLOR, LEGI-CNRS, FABIEN GODEFERD, LMFA-EC Lyon — We present a novel experimental and numerical study on the formation of columnar vortical structures by inertial waves in a rotating fluid. Two inertial-wave cones are generated by a vertically oscillating torus in a fluid in solid body rotation At the tip of the cones, there is a singular point towards which the energy of the waves gets focused. The particularity of this configuration, as compared to those of previous experiments (e.g. oscillating sphere or disc), is that the singular point's position within the fluid leads to complex non-linear wave interaction, which may lead to the formation of a localized vortex that expands in the vertical in the form of a Taylor column. Using detailed PIV measurements we consider the flow evolution from the localized wave overturning motion to the Taylor column formation as well as the inertial wave dynamics during this process, The results are discussed in the context of turbulence in rotating fluids.

[1]We acknowledge financial support from projects ANR ANISO and CIBLE

5:45PM E26.00006 Secondary bifurcation of mixed-cross-spirals (MCS) connecting different travelling wave solutions , SEBASTIAN ALTMEYER, CHRISTIAN HOFFMANN, Institut für Theoretische Physik, Universität des Saarlandes — We investigated numerically in the Taylor-Couette system secondarily bifurcating structures connecting different helical travelling wave solutions with different azimuthal wave numbers, realized by so-called mixed-cross-spirals (MCS), by solving the full Navier-Stokes equations with a combination of a finite difference and a Galerkin method for fixed axial periodicity lengths. Dynamics, stability, and bifurcation behaviour for counter rotating as well as co-rotating cylinders are presented. These secondarily forward bifurcating, stationary flow states - MCS - solutions can be seen as nonlinear superpositions of the involved pure spiral solutions. E.g for a L3R5-MCS. Thereby, the contribution of the respective spiral component (here a L3-SPI and a R5-SPI to the entire MCS varies continuously with the control parameters. In that notation, the well-studied cross-spirals (CSPI) represent a special case of MCS as they consist of two *mirror-symmetric* spiral components. Moreover the ribbon (RIB) solution (axially standing waves) can be seen as an CSPI with equal mode amplitudes (here 3-RIB). Thereby these MCS can appear in two different ways. They can appear as a bypass solution connecting the *same* SPI branch or they can appear as an interim solution connecting *different* SPI branches.

5:58PM E26.00007 Computational simulations of rotating-disk induced flows in circular cylinders , DANIEL T. VALENTINE, MICHAEL J. BRAZELL, BRIAN HELENBROOK, Clarkson University — Numerical solutions of the axisymmetric Navier-Stokes equations are presented that illustrate the flow regime where vortex breakdown occurs. The parameters of the problem are the aspect ratio of the cylindrical container, $A = H/R$, where H is the height of the cylinder and R is the radius of the cylinder, the radius ratio of the radius of the disk, R_d/R, where R_d is the radius of the disk, and the dimensionless rotation speed, viz., the Reynolds number, $Re = R^2\Omega/\nu$, where Ω is the angular velocity of the disk and ν is the kinematic viscosity of the Newtonian fluid within the container. Comparisons with the now classical lid-driven flows are made. Flow visualizations of vortex breakdown in a rotating disk driven flow observed in a relatively crude experimental apparatus is presented to illustrate the robustness of this phenomenon. Comparisons with numerical predictions illustrate that this phenomenon is predominantly an axisymmetric phenomeon. The flow visualizations illustrate interesting three- dimensional features that need further investigation.

6:11PM E26.00008 Optimal Taylor-Couette turbulence[1] , SANDER G. HUISMAN, Physics of Fluids - University of Twente, DENNIS P.M. VAN GILS, SIEGFRIED GROSSMANN, CHAO SUN, DETLEF LOHSE — Strongly turbulent Taylor-Couette flow with independently rotating inner and outer cylinder with a radius ratio of $\eta = 0.716$ is experimentally analysed. From global torque measurements, the maximum in the angular velocity transport from the inner to the outer cylinder is found at slight counter-rotation, namely at an angular velocity ratio of $a_{opt} = -(\omega_o/\omega_i)_{opt} \approx 0.37$. This value is theoretically interpreted and predictions for general η are made. With the help of laser Doppler anemometry, we in addition provide angular velocity profiles. The ratio $a_{opt} \approx 0.37$ is distinguished by zero angular velocity gradient $\partial \langle\omega\rangle_t /\partial r = 0$ in the bulk. For stronger counter- rotation $|\omega_o| > 0.37\omega_i$ the probability distribution function of the bulk angular velocity becomes bi-modal, reflecting intermittent bursts of turbulent structures beyond the neutral line into the outer flow domain, which otherwise is stabilized by the counter-rotating outer cylinder.

[1]This work was financially supported by STW, The Netherlands.

6:24PM E26.00009 Non-Oberbeck-Boussinesq effects in rotating Rayleigh-Bénard convection , SUSANNE HORN, OLGA SHISHKINA, CLAUS WAGNER, German Aerospace Center — We present results from Direct Numerical Simulations of rotating Rayleigh-Bénard convection in a cylindrical cell with unity aspect ratio for water with a Prandtl number of $Pr = 4.38$ and glycerol with $Pr = 2547.9$ with a special focus on temperature dependent material properties and thus not within the framework of the conventionally used Oberbeck-Boussinesq (OB) approximation. The generated turbulent flow fields are analysed with respect to the deviations from the OB cases. For the non-Oberbeck-Boussinesq cases we obtain a breakdown of the top-bottom symmetry, that is different boundary layer thicknesses, asymmetric velocity flow patterns and modified mean temperature profiles including an increase of the centre temperature and in the case of rotation a diminished temperature gradient within the bulk. Furthermore we find a slightly different scaling of the Nusselt number with the Rayleigh as well as with the Rossby number.

6:37PM E26.00010 Heat transport by liquid metal convection with and without rotation , ERIC KING, UC Berkeley, JONATHAN AURNOU, UC Los Angeles — Convection in liquid metals can be quite different than in other fluids such as air or water, due to their small Prandtl numbers. Studying such systems can therefore give us a different perspective on convective turbulence. Furthermore, rotating convection in liquid metal is responsible for the generation magnetic fields on Earth and other planets. We present results from Rayleigh-Bénard convection experiments in liquid gallium with and without rotation. The key measurement of interest is heat transport, and heat transport by liquid metal is contrasted with that by water in a similar system.

Monday, November 21, 2011 8:00AM - 9:57AM –
Session G1 Geophysical Flows: Oceanography II 301

8:00AM G1.00001 Mass Transport by Large Amplitude Internal Solitary Waves[1] , ALAN BRANDT, KARA BLAINE, Johns Hopkins Univ. APL — Mode-2 internal solitary waves (ISW) (symmetric "bulge" waves) have been observed on pycnocline interfaces in the coastal ocean. Mode-2 ISW with sufficiently large amplitudes can have closed streamlines and regions of internal recirculation resulting in entrainment and transport of mass. To investigate the extent of mass transport, a series of laboratory experiments was conducted in a two-layer stratified channel by releasing a volume of fluid at a density equal to that of the interface. The released fluid formed a series (one to three) of large amplitude mode-2 ISW that propagated along the interface. The extent of mass transport was determined by measuring the volume of the initially dyed release fluid contained within the ISW. It was found that the ISW mass transport was governed by the entrainment of external fluid and leakage from the trailing edge. These effects resulted in the larger amplitude ISW having a smaller mass transport decay rate with distance.

[1]Supported by the Office of Naval Research.

8:13AM G1.00002 Internal wave driven transport of fluid away from the boundary of a lake , CHRIS REHMANN, Iowa State University, DANIELLE WAIN, University of Washington, MICHAEL KOHN, JOSHUA SCANLON, Iowa State University — A field study was conducted at West Okoboji Lake near Arnolds Park, Iowa to study transport of fluid resulting from wind forcing on summer stratification. A tracer was injected at the boundary of the lake and surveyed on four separate occasions over a period of three days. Although the Lake number was below 10 in two sustained periods, no elevated mixing occurred near the boundary. Nevertheless, dye injected at the boundary moved almost 950 m into the interior in 29 h. Advection by internal waves accounts for much of the transport, and shear dispersion from the spatially varying velocities in the internal wave field may explain at least some of the rest.

8:26AM G1.00003 Propagating and evanescent internal gravity waves in the deep ocean ,

MATTHEW S. PAOLETTI, M.C. DRAKE, HARRY L. SWINNEY, University of Texas at Austin — We present experimental and numerical studies on internal gravity waves in a stably stratified fluid designed to model the deep ocean. Internal gravity waves generated by tidal flow over ocean topography are responsible for much of the energy transfer in the ocean. King et al. recently found that there exist regions in the deep ocean where the density gradient becomes so small that the buoyancy frequency (proportional to the square root of the density gradient) becomes smaller than the tidal frequency; below such (previously unknown) "turning points" the internal gravity waves become evanescent. The present experiments and simulations examine internal wave reflection and energy transfer at turning points. Further, we study internal wave generation for tidal flow over a ridge on the ocean bottom, and we examine how the evanescent and propagating internal wave intensities depend on the height of the ridge relative to the turning point depth.

8:39AM G1.00004 Large amplitude internal waves in weakly stratified oceans ,

WOOYOUNG CHOI, NJIT, ROXANA TIRON, KAIST, ROBERTO CAMASSA, UNC-CH — We consider large amplitude internal waves in weakly stratified fluids and obtain the nonlinear evolution equations, which generalize the strongly nonlinear model for a system of two constant density layers. After the linear dispersion relation of the new model is compared with that of the linearized Euler equations, the solitary wave and conjugate state solutions of the model are obtained and compared with other theoretical solutions and field data.

8:52AM G1.00005 Internal wave–vorticity coupling for an oscillating disk[1] ,

BRUNO VOISIN, LEGI, CNRS & University of Grenoble, SYLVAIN JOUBAUD, THIERRY DAUXOIS, ENS de Lyon, CNRS & University of Lyon — In a density-stratified fluid, viscosity couples internal waves with vertical vorticity. So far this coupling used to be neglected in analytical studies and only the viscous attenuation and spreading of the waves was taken into account, except in a very recent study of the oscillations of a horizontal circular disk.[2] We investigate the relations between the previous analytical approaches of the disk, considering either inviscid or viscous propagation of the waves and either free- or no-slip conditions at the disk, and compare their output with an original approach based on the boundary integral method. In particular, the role of the Stokes number is clarified. The analytical predictions are compared with contact measurements for vertical oscillations[3] and with original PIV measurements and visualizations for both vertical and horizontal oscillations.

[1]Supported by grant PIWO of the ANR (France).

[2]A. M. J. Davis & S. G. Llewellyn Smith, J. Fluid Mech. **656**, 342–359 (2010).

[3]R. N. Bardakov, A. Yu. Vasil'ev & Yu. D. Chashechkin, Fluid Dyn. **42**, 612–616 (2007).

9:05AM G1.00006 ABSTRACT WITHDRAWN —

9:18AM G1.00007 Boluses leading the run up of shoaling internal waves of elevation ,

MICHELLE E. PEDE, DANIEL T. VALENTINE, Clarkson University — Numerical solutions of the two-dimensional Navier-Stokes and convective-diffusion equations, to within the Boussinesq approximation, are presented. These solutions illustrate the mechanism of production, ejection and propagation of boluses, i.e., vortex projectiles, induced by shoaling internal solitary waves of elevation. They are the type of boluses observed in laboratory and field experiments reported in the geophysical fluid dynamics literature. The boluses predicted in this study are *not* a consequence of a breaking event. They are the shedding of a packet of fluid with concentrated vorticity that forms a vortex. This vortex is formed as the leading portion of the shoaling internal solitary wave of elevation steepens. The wave of elevation contains a significant concentration of vorticity. This vorticity is accumulated near the front of the wave by a combination of convection and production by enhancement of horizontal gradients of density as it shoals. The concentration of vorticity forms a vortex projectile. It interacts with its image on the other side of the shoal in the same way two vortices of opposite strength interact. The image vortex system propels the vortex projectile up the shoal a relatively large distance, even as part of the incident wave reflects and recedes offshore. The bolus leading the run-up process contains fluid that is heavier than the surrounding fluid and, hence, transports bottom layer fluid quite effectively to locations far up the sloped topography.

9:31AM G1.00008 Exact solutions for waves over periodic topography of arbitrary shape and amplitude[1] ,

JIE YU, Department of Civil, Construction and Environmental Engineering, North Carlonia State University — Understanding the interaction of wave and seabed topography is important, due to its relevance to sediment transport and wave transformation in coastal oceans. For periodic topography of large amplitude, exact theoretical solutions are few, even for linear waves. Recently, Howard and Yu (2007, J. Fluid Mech.) have developed an exact theory for linear waves over large amplitude bottom corrugations, using a conformal transformation. A complete set of Floquet-type solutions is given, which are analogous to the families of propagating and evanescent modes over the flat bottom. The corrugations considered in Howard and Yu have a shape which is approximately sinusoidal at small amplitude and becomes increasing cusp-like as the amplitude increases. Here, we have extended the theory of Howard and Yu to consider periodic topographies of arbitrary shape and amplitude. Examples are shown for purely sinusoidal, doubly sinusoidal bottom undulations, and square-wave like bottom topography.

[1]This work is in collaboration with Louis N. Howard (MIT). JY is supported by National Science Foundation (Grant CBET-0845957).

9:44AM G1.00009 The Effect of Wave Intensity on Fine Sediment Transport in Oscillatory Channel ,

CELALETTIN OZDEMIR, TIAN-JIAN HSU, Civ. & Env. Eng. University of Delaware, S. BALACHANDAR, Mech. & Aerosp. Eng University of Florida — Simulations on particle-laden flow in oscillatory channel are critical to further understand suspension and offshore delivery of fine terrestrial sediment in the coastal ocean. In an effort to understand these mechanisms, the degree of sediment induced density stratification quantified by Richardson number, Ri and settling velocity of particles, V_s, have been studied through turbulence resolving simulations with simplified Eulerian-Eulerian two phase model (Ozdemir et al. 2010, 2011). In these studies, moderately energetic wave field is considered given by Stokes Reynolds number, Re, of 1000 where all the scales of turbulence can be resolved. As a result, existence of four regimes of particle transport has been identified and presented in the parametric space defined by V_s and Ri. From low to high Ri (or V_s), these regimes range from well-mixed condition to the formation of lutocline, and eventually a complete flow laminarization. Here, we further demonstrate the impact of lower wave intensity (Re) to the modes of sediment transport. Results show significant changes in the particle transport modes under variable V_s and Ri. The culmination of the results so far yields a comprehensive picture on the nature of transitional turbulence due to different degree of sediment impact and the mode of mixing and transport in the oscillatory channel.

Monday, November 21, 2011 8:00AM - 10:10AM —

Session G2 Nanofluids II 302

8:00AM G2.00001 Continuum and Molecular Dynamics Studies On the Diffusiophoretic Motion , NIMA SHARIFI-MOOD, JOEL KOPLIK, CHARLES MALDARELLI, the City College of New York & Levich Institute — The self-propulsion of micron or sub-micron objects has a number of applications as miniaturized motors. One method for particle self propulsion is to utilize a surface chemical reaction on one part of the particle surface to create concentration gradients of solutes across the particle. These gradients drive a diffusiophoretic motion due to unbalanced (van der Waals) attractions between the particle and the solutes and solvent within an intermolecular length scale (L, 10-100 nm) of the particle surface. Prior continuum studies assume the interaction creates a local slip velocity at the particle surface, and find the terminal velocity U of spherical particles to be independent of the radius a. We provide numerical solutions for U which account directly for the solute transport and flow within L, and matched asymptotic solutions as L/a tends to zero. The leading order expression for U is independent of a, but U decreases with the particle radius for L/a greater than .01. Molecular dynamics simulation is also undertaken using Lennard-Jones potentials to provide a more complete picture of nanoscale propulsion.

8:13AM G2.00002 Nanofluid heat transfer anomaly: Theoretical explanation of observations in the development region of microchannels , J.T.C. LIU, School of Engineering, Brown University — Continuing from http://meetings.aps.org/link/BAPS.2010.DFD.CP.7 is the work on analytical solutions to the first order perturbation problem for momentum, heat and volume concentration following the continuum conservation equations for nanofluids (Buongiorno 2006; Pfautsch 2008; Tzou 2008), simplified by the Rayleigh-Stokes approximation and perturbation in small free stream volume fraction. The disparate momentum, heat and volume fraction transport layer thicknesses$\delta_u > \delta_T >> \delta_\phi$, are estimated for metallic nanofluids, further structures the transport problems. From experiments of, e.g., Wen & Ding 2004, Jung, et al. 2009, it is concluded that the observed large "anomalous" surface heat transfer rates for small increases in the volume fraction, especially at the leading edge of laminar microchannel nanofluid flows, is partially attributable to the nanofluid modification of the temperature profile by the inertial effects of modified density and heat capacity and of conduction effects of the modified thermal conductivity. The solutions obtained display these contributions explicitly. Thus the use of nanofluid transport properties in the correlation of laminar heat transfer must necessarily be accompanied by the detailed considerations of the temperature profile modification in nanofluid flow via the conservation equations.

8:26AM G2.00003 Transport of nonaparticles flowing past a patterned substrate[1] , RUI ZHANG, JOEL KOPLIK, City College of CUNY — We consider the defection of suspended particle trajectories due to flow past a patterned surface using complimentary Langevin numerical simulations and theoretical analyses based on the Fokker-Planck equation. The intended application is to vector particle separation in a nanofluidic channel. The simulations of flow past an alternating periodic striped pattern of attractive and repelling regions generally show that particles are deflected away from the imposed flow direction, to a degree that depends on the particle's size and the details of the substrate interaction. Both van der Waals and Coulomb interactions are studied, and in the latter case we explore the effects of the spatial distribution of bounding surface charge. Theoretically, both exactly in two dimensions and within the Fick-Jacobs approximation in three, we show that the effect of a periodic potential on the particle motion is always to impede the mean transport velocity in the orthogonal direction and deflect the trajectory, or equivalently reduce the effective diffusivity. Upper and lower bounds are found for the deflection angle, and explicit estimates are obtained in the limit of a weakly varying interaction.

[1]Supported by the NSF.

8:39AM G2.00004 Molecular dynamics simulations of Janus nanoparticles in evaporating droplets , WEIKANG CHEN, JOEL KOPLIK, ILONA KRETZSCHMAR, City College of CUNY — We use numerical simulations to study the evaporation of sessile droplets containing Janus particles placed on a heated surface. The droplets are composed of a Lennard-Jones fluid, the particles are rigid spherical sections of an atomic lattice and heating is controlled through the temperature of the (atomistic) surface. The study focuses on the time evolution of the droplets shape, contact angle, velocity field and evaporative flux, as well as the motion of the particles and the effects of the Janus interactions on the structure of the particulate deposit. The results are compared with recent theory and experiment.

8:52AM G2.00005 Properties of Water/Gold Nanofluids[1] , GIANLUCA PULITI, SAMUEL PAOLUCCI, MIHIR SEN, University of Notre Dame — Nanofluids belong to a new class of fluids with enhanced thermophysical properties and heat transfer performance. A broad spectrum of applications in science and engineering can potentially benefit from their use. However, the physical explanation for this enhancement is still lacking. The novelty of this work is in a fundamental, realistic, and comprehensive approach to the problem of understanding nanofluids through the use of molecular dynamics simulations with accurate potentials to model realistic materials. Specifically, this study treats the case of water confined between gold nanolayers to examine interfacial interactions and a water-based nanofluid with spherical gold nanoparticles. Thermodynamics and transport properties will be discussed for both systems. It is interesting to note that while the thermodynamic properties of the mixture are typically predicted using ideal mixture theory, such predictions are found to be generally poor for nanofluids. The anisotropy induced by the gold-water interface, and its effects appear to be responsible for the disagreement. We will also discuss the role of interfacial effects on the anomalous enhancement of nanofluid transport properties.

[1]Funding for this research is provided by DOE grant FG02-05-CH11294.

9:05AM G2.00006 A comparative study of different ferrofluid constitutive equations. , PURNA KALONI, University of Windsor — Ferrofluids are stable colloidal suspensions of fine ferromagnetic monodomain nanoparticles in a non-conducting carrier fluid.The particles are coated with a surfacant to avoid agglomeration and coagulation.Brownian motion keeps the nanoparticles from settling under gravity.In recent years these fluids have found several applications including in liquid seals in rotary shafts for vacuum system and in hard disk drives of personal computers,in cooling and damping of loud speakers, in shock absorbers and in biomedical applications. A continuum description of ferrofluids was initiated by Neuringer and Rosensweig [1] but the theory had some limitations. In subsequent years,several authors have proposed generalization of the above theory.Some of these are based upon the internal particle rotation concept, some are phemonological,some are based upon a thermodynamic framework,some employ statistical approach and some have used the dynamic mean field approach.The results based upon these theories ane in early stages and inconclusive. Our purpose is, first, to critically examine the basic foundations of these equations and then study the pedictions obtained in all the theories related to an experimental as well as a theoretical study. [1]J.L.Neuringer and R.E. Rosensweig, Physics Fluids,7.1727 (1964)..

9:18AM G2.00007 Influence of Red Blood Cells on Nanoparticle Targeted Delivery in Microcirculation , JIFU TAN, ANTONY THOMAS, YALING LIU, Lehigh University — In this paper, a particle-cell hybrid model is developed to model Nanoparticle (NP) transport, dispersion, and binding dynamics in blood suspension under the influence of Red blood cells(RBCs). The motion and deformation of RBCs is captured through the Immersed Finite Element Method. The motion and adhesion of individual NPs are tracked through Brownian adhesion dynamics. A mapping algorithm and an interaction potential function are introduced to consider the cell-particle collision. NP dispersion and binding rates are derived from the developed model under various rheology conditions. The influence of RBCs, vascular flow rate, and particle size on NP distribution and delivery efficacy is characterized. A non-uniform NP distribution profile with higher particle concentration near the vessel wall is observed. Such distribution leads to over 50% higher particle binding rate compared to the case without RBC considered. The tumbling motion of RBCs in the core region of the capillary is found to enhance NP dispersion, with dispersion rate increases as shear rate increases. Results from this study contribute to the fundamental understanding and knowledge on how the particulate nature of blood influences NP delivery, which will provide mechanistic insights on the nanomedicine design for targeted drug delivery applications.

9:31AM G2.00008 The Role of Hydrodynamic Behavior of DNA Molecules in Dielectrophoretic Polarization under the Action of an Electric Field[1] , HUI ZHAO, University of Nevada Las Vegas — A continuum model is developed to predict the dielectrophoretic polarizability of coiled DNA molecules under the action of an alternating current electric field. The model approximates the coiled DNA molecule as a charged porous spherical particle. The model explains the discrepancies among scaling laws of polarizability of different-sized DNA molecules with contour length and such discrepancies are attributed to different hydrodynamic behavior. With no or one fitting parameter, theoretical predictions are in good agreements with various experimental data, even though in experiments there are some uncertainties in regards to certain parameters.

[1]This work was supported, in part, by UNLV URBAN 21 Innovation Grant.

9:44AM G2.00009 Emission and Charging of Nanoaerosol Plumes from a Taylor Cone-Jet , YUNSHAN WANG, Dept. of Chemical and Biomolecular Engr., Univ. of Notre Dame, MING K. TAN, Dept. of Chemical and Biomolecular Engr. and Dept. of Aerospace and Mechanical Engr., Univ. of Notre Dame, DAVID B. GO, Dept. of Aerospace and Mechanical Engr., Univ. of Notre Dame, HSUEH-CHIA CHANG, Dept. of Chemical and Biomolecular Engr., Univ. of Notre Dame — We examine the explosive atomization at the tip of a Direct-Current Taylor cone-jet for an electrolyte, which is the most common mode for electrospray proteomic mass spectrometry applications but whose fundamental mechanism remains unknown. With scaling arguments and imaging experiments, we demonstrate the underlying physics to be the induction of a polarized region with a high space charge density in the microjet by the dominant Taylor harmonic of the cone. The induced space charge density increases along the jet until Coulombic repulsion among the space charges, when their separation is below the Bjerrum length, triggers the explosive microjet atomization and emits charged nanoaerosols in distinct conic plumes. As the induced microjet space charge can be estimated by a nonlinear Guoy-Chapman equilibrium analysis for strong electrolytes, the jet breakup length, current-flow rate correlation, aerosol size/charge and plume angle are strong functions of ionic strength and interfacial tension and are quantitatively captured by our universal scaling laws.

9:57AM G2.00010 Computer simulation of DNA translocation in functionalized conical nanopores[1] , GUOQING HU, LNM, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, BAO JIAO — As a powerful research tool to rapidly investigate the dynamic and structural properties of single biomolecules such as DNA and protein, solid-state single nanopores have attracted a great deal of scientific interests. In a typical setup, DNA molecule is driven electrokinetically and its translocation is conveniently detected via the ionic current measurement through single nanopores. One of challenges arising from solid-state nanopore DNA sequencing technique is the molecular selectivity and detection resolution. Tuning the geometry and surface property of nanopores helps create sensing devices that control transport of ions and molecules in electrolyte. Here we present molecular dynamics simulation method to study the ion transport and DNA translocation phenomena in conically shaped nanopores coated with single-stranded DNA molecules. Computer simulations characterize the effects of the effective nanopore diameter, conical pore geometry, ionic strength, surface charge, and applied biasing voltage on the ionic current and DNA translocation dynamics. The findings can be used to improve and optimize the experimental design for DNA detection in single nanopore devices.

[1]Support from the National 973 Project in China (Grant No. 2011CB707604) is acknowledged.

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G3 Surface Tension II 303

8:00AM G3.00001 Using bulk convection to approach kinetic-limited surfactant dynamics , LYNN WALKER, NICOLAS ALVAREZ, Carnegie Mellon University, DOUGLAS VOGUS, Bucknell University, SHELLEY ANNA, Carnegie Mellon University — Many processes involving transport of surfactants to interfaces are in the regime where kinetics, diffusion and convection are comparable. Using the principle that the timescale for diffusion depends on curvature, we previously developed a microtensiometer to accurately measure surfactant dynamics at spherical microscale liquid-fluid interfaces. In the present study, we introduce a low Reynolds number flow in the bulk solution to further increase the rate of diffusion. Dynamic surface tension is measured as a function of Peclet number and the results are put into the context of a simplified convection-diffusion model. Although a transition from diffusion to kinetic- limited transport is not observed experimentally for the surfactants considered, lower bounds on the adsorption and desorption rate constants are determined. These lower bound values are much larger than previously reported rate constants. This experimental tool and analysis allows the governing mechanisms of surfactant transport at liquid-fluid interfaces to be quantified. The addition of flow near an interface is effective in decreasing the length scale for diffusion, and shifting the relevant timescales. The results show that the detailed nature of the flow field does not need to be controlled as long as the local Reynolds number is low.

8:13AM G3.00002 Spinodal decomposition in particle-laden Landau-Levich flow , JUSTIN KAO, A.E. HOSOI, Massachusetts Institute of Technology — We investigate Landau-Levich coating of a solid wall by a suspension of large spherical particles. Capillary forces lead to self-assembly of a monolayer of particle aggregates, and experiments show two regimes of pattern formation, corresponding to sporadic and continuous deposition. We obtain coating fraction as a function of bulk particle volume fraction and wall speed, and propose a continuum spinodal decomposition (Cahn-Hilliard) model for this pattern formation process. Solutions of the corresponding model equations are presented and compared to experimental results.

8:26AM G3.00003 Nanoparticle monolayers under stress: mechanically forced desorption from a fluid-fluid interface , VALERIA GARBIN, JOHN C. CROCKER, KATHLEEN J. STEBE, University of Pennsylvania — Nanoparticle-laden interfaces are studied for applications to materials with tunable electronic and optical properties, as emulsion stabilizers, and in catalysis. The mechanical response of nanoparticle monolayers under applied stress is of emerging interest since it impacts the success of these applications. Here we focus on the response of nanoparticle-laden interfaces to compression. A monolayer of nanoparticles is allowed to spontaneously form by adsorption from an aqueous suspension onto a pendant drop of oil. The effective surface pressure Π of the composite interface is monitored by pendant drop tensiometry. As the drop is compressed, the nanoparticles are mechanically forced out of the interface into the aqueous phase. A new optical method is developed to measure the nanoparticle area density *in situ*. We show that desorption occurs at a coverage that corresponds to close packing of the ligand-capped particles, suggesting that ligand-induced repulsion plays a crucial role in the desorption process.

8:39AM G3.00004 The effect of lipid monolayers on Faraday waves[1] , STEPHEN STRICKLAND, Dept. of Physics, NC State University, LAKE BOOKMAN, MICHAEL SHEARER, Dept. of Mathematics, NC State University, KAREN DANIELS, Dept. of Physics, NC State University — Surface tension is known to affect the critical driving acceleration for Faraday waves and their spatial wavenumber at onset. We perform experiments in the subharmonic regime, on water whose free surface is contaminated with up to one monolayer of fluorescent NBD-PC lipid. A circular container of water is vibrated vertically at single frequencies ranging from 15 Hz to 70 Hz, and we measure the acceleration and wavenumber at the onset of Faraday waves. We observe that the critical acceleration is larger than predicted by recent models, if the effect of the contaminant is assumed to simply lower the surface tension. Critical wavenumbers are largely unaffected. We examine whether a non-uniform lipid distribution is responsible for these effects.

[1]This work is funded by NSF Grant # DMS-0968258.

8:52AM G3.00005 Surfactant-Driven Fracture Formation in Soft Gels[1] , MARK SCHILLACI, Dept. of Physics, NC State University, JOSHUA BOSTWICK, Dept. of Mathematics, NC State University, KAREN DANIELS, Dept. of Physics, NC State University — The formation of fractures in gels well above the solid-liquid transition has been previously shown to initiate through a Poisson process, indicating that thermal fluctuations play a significant role. Here, we present experiments quantifying fracture formation in gels close to the solid-liquid transition. We utilize a spreading surfactant droplet to apply small forces to the surface of the gel. Fractures form along the contact-line and propagate outward in a star-burst pattern. By varying the droplet surface tension and gel modulus, we are able to tune the fracture formation and control the mean number of fractures formed. We interpret the number of fractures formed in the context of a linear elastic model for the uncompensated, Young-Dupré (out-of-plane) force acting at the contact-line. However, we also observe that there is an inherent variability in both the number of fractures formed and the delay for fractures to form. In the regime where single fractures form, we observe a range of delay times consistent with a Poisson distribution. In the regime where multiple fractures form, we observe that all fractures appear simultaneously and the long delays are suppressed.

[1]This work was supported by NSF grant number DMS-0968258.

9:05AM G3.00006 Rod-like microparticles at interfaces: near-field capillary interactions and implications for mechanics of particle-laden interfaces , LORENZO BOTTO, LU YAO, MARCELLO CAVALLARO, KATHLEEN J. STEBE, Chemical and Biomolecular Eng. Dept. University of Pennsylvania, STEBE'S LAB TEAM — Rod-like particles assemble at fluid interfaces owing to anisotropic capillary interactions. Experiments on cylindrical and ellipsoidal particles reveal that preferred orientation, interaction strength, and properties of resulting assemblies depend strongly on particle shape. While cylinders assemble end-to-end forming rigid linear chains, ellipsoids form flexible structures with particles side-by-side. Simulations of pairs of particles at contact reveal that chains of cylinders are rigid. Experiments bear this out: a chain of microcylinders rotated via a magnetic field remains rigid even under torques in excess of $10^5 kT$. Above a yield stress, chains snap and dissipate stored capillary energy. Interactions between ellipsoids are comparatively weak, with no energy barrier as particle rotate about each other, consistent with the highly flexible chains formed by these particles. These interactions have profound implications for the mechanics of particle-laden interfaces including their viscoelastic properties.

9:18AM G3.00007 Experimental and Theoretical Study of Chemically Driven Waves at an Oil-Water Interface , MEIR BASSON, ROUSLAN KRECHETNIKOV, University of California at Santa Barbara — When an aqueous solution of TSAC surfactant is placed on top of a heavier organic solution of Iodine, a chemical reaction between the solutes causes traveling waves to form at the interface. These traveling waves have wavelengths on the order of centimeters and speeds on the order of centimeters per second. In this talk, we present an experimental study of these chemically driven interfacial waves that form and rotate around an annular container. Specifically, using stereo photography, we resolve the three dimensional structures and time scales of the waves and thus explore the effect of parameters such as container geometry (e.g. diameter, annular gap) and solute concentrations on the speed, shape, and lifetime of the formed interfacial waves. Also, a theoretical study describing the mechanism behind the traveling waves will be presented. In particular, we analyze the chemical reaction which causes the wave motion and propose a model for the chemical-to-mechanical conversion of energy at the liquid-liquid interface.

9:31AM G3.00008 Thermocapillary Actuation of Binary Drops on Solid Surfaces , FANGJIE LIU, YUEJUN ZHAO, CHUAN-HUA CHEN, Duke University — Under thermocapillary actuation, aqueous drops typically remain stuck on hydrophobic solid surfaces subjected to a temperature gradient. We report the thermal actuation of water drops on a parylene-coated silicon substrate, where the actuation was enabled by encapsulating the water drop with a minute amount of long-chain alcohol. When placed near an initially stuck water drop, heptanol spread toward the water under thermocapillarity. The merged binary drop can take on a unique shape: the dome shaped water drop is capped by a thin layer of heptanol and the cap protrudes through a "foot" to a precursor film. For intermediate volumes of water drop with a fixed volume of heptanol, the speed of the binary drop was linearly proportional to its diameter and the imposed temperature gradient with an offset accounting for the hysteresis force. The thermocapillary actuation speed can be modeled by a lubrication theory adapted from the models of slender liquid ridges. The binary thermocapillary actuation also works for other combinations of secondary fluids and solid surfaces, making it an attractive scheme for transporting aqueous samples of biomedical and industrial relevance.

9:44AM G3.00009 Shape oscillations of attached bubbles in pure liquids and in surfactant solutions[1] , JIRI VEJRAZKA, LUCIE VOBECKA, NICOLAS MEUNIER, MARIA ZEDNIKOVA, SANDRA ORVALHO, JAROSLAV TIHON, Academy of Sciences of the Czech Republic — The high-speed imaging is used for studying oscillations of air bubble, which are excited by a motion of a tip of a capillary, to which the bubble is attached. The frequency, damping and shape of decaying oscillations are determined for the lowest three eigenmodes. In pure water, the experimental results agree well with an inviscid analysis of Bostwick and Steen (*Phys Fluids* **21**, 2009), if the bubble size is small compared to the detachment size. For larger bubbles, the effect of neck formation on the oscillations is documented experimentally. The addition of a surfactant to the system modifies both the oscillation frequency and damping. The frequency initially increases with the increasing surfactant concentration due to Gibbs elasticity, and starts to decrease only after exceeding some surfactant concentration. The damping strongly increases, passes through a maximum and then slightly decreases. The maximum damping is observed for the concentration, at which the oscillation frequency starts to decrease. Concluding, bubble oscillations are strongly sensitive to presence of a surfactant.

[1]Research supported by Czech Science Foundation (P101/11/0806).

9:57AM G3.00010 Solid Layer Formation at Oil-Water Interfaces , RIELLE DE RUITER, WILLEM TJERKSTRA, MICHEL DUITS, FRIEDER MUGELE, Univ. Twente, Physics of Complex Fluids — Metal stearates form at the interface between a decane solution of stearic acid (s.a.) and aqueous salt solutions of variable composition and pH. Studying the evolution of their mechanical, optical, and chemical properties as a function of time we find hardly any interfacial activity for pH<pKa of s.a. For pH > pKa, s.a. deprotonates at the interface and forms metal stearates, eventually leading to the formation of macroscopic solid layers. Dynamic interfacial tension measurements reveal several stages of the process, including the subsequent formation of dilute and dense monolayers followed by three-dimensional growth. In the presence of divalent ions, the solid layers display a significant increase in the dilatational storage modulus. In the presence of multiple cation species (artificial seawater) the growth of the solid layers is particularly pronounced. The layers preferentially incorporate Ca2\th as revealed by XPS and IR spectroscopy. Our results highlight in particular the importance of the synergistic effects of simultaneously present monovalent and divalent cation species on the interfacial adsorption.

Monday, November 21, 2011 8:00AM - 10:10AM – Session G4 Drops IV: Breakup 307

8:00AM G4.00001 Breakup of liquid filaments over a wide range of Ohnesorge number[1] , ALFONSO A. CASTREJON-PITA, JOSE R. CASTREJON-PITA, IAN M. HUTCHINGS, University of Cambridge — This work presents a study of the breakup of liquid filaments over a wide range of Ohnesorge numbers (0.001 to 10) using a simple large-scale jet generator. The experimental arrangement features a variable-size nozzle, whose radius can be adjusted from 100 μm to 3 mm and which is capable of jetting liquids with viscosities between 1 mPa s and 1.4 Pa s. The actuator of this generator consists of an electromagnetic vibrator that can be driven by arbitrary waveforms in order to control the jet formation process. The instrumentation also includes a fast pressure transducer to monitor the true pressure drive produced by the actuator. The filament breakup behavior and distribution (regime diagram), in terms of aspect ratio and Ohnesorge number, is compared with the predictions from previously published models.

[1]This project was supported by the EPSRC and industrial partners in the Glassjet and I4T projects.

8:13AM G4.00002 ABSTRACT WITHDRAWN —

8:26AM G4.00003 Instability of the crown formed by a splashing droplet , XINJUN GUO, Department of Physics, Brown University, SHREYAS MANDRE, School of Engineering, Brown University — Splashing from impacts between liquid drops and a surface is of importance for many natural and industrial processes. The mechanism which dominates the crown breakup has mostly been proposed to be the capillary instability of the rim that forms and grows at the boundary of the crown. But no such theory has so far captured the main properties of the crown. Rayleigh's theory of capillary thread breakup predicts the size of secondary drops to be comparable with the thickness of the unperturbed rim, but the rim is constantly growing. Motivated by this puzzle, we study the instability of a cylindrical thread of liquid, which is constantly growing due to a source of liquid along its centerline. This is our idealized model for the breakup of a liquid lamella rim into droplets. We find that the most unstable wavelength grows with time proportionally to the radius of the thread and the amplitude of the perturbations do not grow like $e^{\lambda t}$ but like $e^{\lambda t^{1/4}}$. Also, the stability theory we use on this time-dependent non-autonomous system generalizes the traditional modal stability theory and the non-modal approaches that have developed more recently.

8:39AM G4.00004 Pinch-off and reconnection of axi-symmetric drops in Stokes flow , SHADI NADERI, MONIKA NITSCHE, University of New Mexico — Under certain conditions, a drop in Stokes flow subject to a non-linear strain field collapses at two points on the axis, leading to a finite time pinch-off singularity. The evolution towards pinch-off is computed using a high order method with local mesh refinement, and found to satisfy a linear self-similar scaling law, as expected. A numerical method is presented to simulate the breaking of the drop at pinch-off and its reconnection into three pieces. We verify that the drop evolution after reconnection is not affected by details of the numerical method. The post reconnection recoiling process is found to satisfy a linear scaling law as well. The computed evolution of the middle drop is furthermore observed to display pinch-off at least two more times.

8:52AM G4.00005 Gas-Liquid Droplet Microfluidics , CARLOS HIDROVO, BRIAN CARROLL, The University of Texas at Austin — The possibility of replacing the highly viscous continuous phase oil for a less viscous gaseous phase offers opportunities for higher flow rates, reduced pumping power, and increased droplet inertia for high speed mixing applications. Liquid droplet generation in a gaseous microflow, however, is often characterized as unwieldy, difficult, and less forgiving compared to aqueous droplet generation in silicon oils. Creating droplets in common microchannel geometries, such as T-junctions and flow focusing arrangements, is undoubtedly possible but the metamorphosis of topology, such as spherical droplets, asymmetrical slugs, trailing pears, and liquid threads is significantly different than liquid-liquid flows. This presentation addresses the fundamental operation of liquid droplet generation in a confined gaseous microflow. Droplet volume, shape, and generation rates are experimentally characterized for common and not-so common microchannel geometries. Transitions from slugs to pools to films for T-junctions and dripping to jetting to threading for flow focusing are identified. High speed images are used to quantify the discrete phase characteristics and qualify the generation and detachment process.

9:05AM G4.00006 A mesoscale study of shear pinch-off , MARCO ARIENTI, Sandia National Laboratories, XIAOYI LI, United Technologies Research Center — The dynamic behavior of a liquid thread undergoing thinning and pinch-off under stretching is simulated using Many-body Dissipative Particle Dynamics (MDPD). The mesoscale nature of this method is first verified with the well-known capillary pinch-off dynamics, where the cascade of self-similar regimes – inviscid (2/3 power law), inertial-viscous (linear power law), and stochastic (0.418 power law) – can be fully captured in a single simulation. Computationally, the imposition of axial stretching requires a new boundary treatment because periodicity cannot accommodate opposing motions at the two ends of the liquid thread. A new algorithm implementing multi-component non-periodic boundary conditions (MCNPBC) is shown to enforce a prescribed liquid-gas interface at the boundary while maintaining a constant number density of both components. The first feature makes multi-component particle methods more flexible, since the computational boundaries can cut through the liquid thread; the second is crucial because in MDPD the repulsive interaction is density-dependent. With this boundary treatment, the strain rate due to an outer fluid is introduced as a parameter in the balance of capillary, viscous, and inertial forces leading to pinch-off.

9:18AM G4.00007 Slender body theory for tip streaming: The formation of concentrated emulsions composed by micron-size drops , ELENA CASTRO-HERNANDEZ, F. CAMPO-CORTES, J.M. GORDILLO, Área de Mecánica de Fluidos, E. T. S. I., Universidad de Sevilla — Under creeping flow conditions a liquid of viscosity μ_i is injected through a cylindrical capillary tube immersed into an outer immiscible fluid with a viscosity μ_o. The outer fluid flows parallel to the axis of the tube. If the outer capillary number $Ca_o > 1$ and the inner to outer flow rate ratio $Q_i/Q_o \ll 1$, the interface exhibits a cone-jet transition similar to that observed in electrosprays. The jet emanating from the tip of the cone is so small that the drops formed can be several orders of magnitude smaller than the diameter of the injector. We present a slender body theory that provides a third order ordinary differential equation for the shape of the interface. The theoretical shape, which is found by shooting with just a single parameter from far downstream towards the tube exit, faithfully reproduces the cone-jet transition observed in experiments for arbitrary values of the three dimensionless numbers that control this physical situation: Ca_o, μ_i/μ_o and Q_i/Q_o. This theory is a powerful design method since it can be used to check the efficiency of new geometrical designs to generate monodisperse emulsions, avoiding the need to build and test them in the laboratory. Drop size is well predicted by the classical Tomotika's stability analysis.

9:31AM G4.00008 Stabilizing toroidal and higher-genus droplets using viscoelastic media , ALBERTO FERNANDEZ-NIEVES, EKAPOP PAIRAM, Physics. Georgia Tech — We inject a viscous liquid through a needle into another rotating viscous liquid to generate toroidal droplets. These droplets are unstable and undergo a transformation into spherical droplets driven by surface tension: They either break ala Rayleigh-Plateau or grow fatter to become a single spherical droplet depending on the aspect ratio of the torus. By replacing the outer phase with a viscoelastic fluid with a non-zero yield stress we can stabilize these and other non-zero genus droplets. We will examine this stabilization mechanism and present criteria to effectively prevent the break-up of these droplets.

9:44AM G4.00009 The hydrodynamics of splash-cup seed dispersal
, GUILLERMO AMADOR, YASUKUNI YAMADA, DAVID HU, Georgia Institute of Technology — Over 20 species of plants have raindrop-sized flowers that catch raindrops opportunistically, directing the resulting splash to scatter their 0.3 mm seeds over distances of 1 meter. In this combined experimental and theoretical study, we measure the effectiveness of this drop-based projectile launching. High speed videography is used to visualize drop impacts onto the conical flowers of the plants *Chrysosplenium* and *Mazus* as well as their shape mimics fabricated using a 3D printer. We observe dispersal distance is strongly dependent on the impact parameter between the drop and flower. Off-center impacts maximize dispersal distance by amplifying fluid velocity by a factor of three in a manner similar to the oblique impact of a jet onto a horizontal plane (Kate et al 2007). Further increase in dispersal distance is accomplished through increase of the inertia of the seed by encapsulation into drops.

9:57AM G4.00010 Mechanical and dynamical responses of pinned drops and polymer balloons
, CHUN-TI CHANG, SUSAN DANIEL, PAUL STEEN, Cornell University — Surface tension is responsible for the nonlinear response of individual spherical-cap drops. This nonlinearity determines the aggregate behavior of a system of interacting drops. Potential applications for droplet systems include switchable-adhesion devices and micro-dosing of personalized pharmaceutical drugs. For a single spherical-cap drop, the pressure-volume (pV) response exhibits one maximum. With the coverage of its free surface by a soft polymer film, the drop becomes a polymer balloon and can change its stability as a result of the reinforced surface tension. The appearance of a second turning point, for example, increases considerably the complexity of the system behaviors exhibited. In this study, we contrast the static and dynamical responses of drops and polymer balloons, taking the perspective that their nonlinear behaviors fall into 1-parameter continuum of pV-responses. Behavior observed in experiment for both excited and unexcited situations will be reported.

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G5 CFD III: Multiphase Flow 308

8:00AM G5.00001 The Immersed Interface Method for Two-Fluid Problems
, MIGUEL UH, SHENG XU, Southern Methodist University — Many problems of fluid mechanics involve the interaction of two immisible fluids. It is generally difficult and inefficient to simulate each fluid separately using an interface-fitted grid method. In the immersed interfaced method a two fluid problem is formulated as one set of governing equations and simulated on a fixed Cartesian grid. The effect of the two-fluid interface enters the formulation as a singular force and a numerical scheme as jump conditions. In these talk we will present the principal jump conditions, discuss the difficulties in implementing them and provide a few options to overcome the difficulties. Finally we will demonstrate the accuracy and efficiency of our immersed interface method for two fluid flow simulations.

8:13AM G5.00002 Using particles to improve gradient-augmented level set methods for surface advection
, OLIVIER MERCIER, J.-C. NAVE, McGill University, R.R. ROSALES, MIT, B. SEIBOLD, Temple University — Level set methods use the zero contour of an implicit function to represent a surface. Traditional methods only track values of the level set function on grid points at each time step. Gradient-augmented methods (and more generally jet-schemes) also keep track and use derivative information. In this talk we will show how these gradient-augmented methods offer a natural framework for incorporating Lagrangian particle information to improve the conservation of mass during advection of surfaces.

8:26AM G5.00003 The Augmented Fast Marching Method for Level Set Reinitialization[1]
, DAVID SALAC, University at Buffalo - SUNY — The modeling of multiphase fluid flows typically requires accurate descriptions of the interface and curvature of the interface. Here a new reinitialization technique based on the fast marching method for gradient-augmented level sets is presented. The method is explained and results in both 2D and 3D are presented. Overall the method is more accurate than reinitialization methods based on similar stencils and the resulting curvature fields are much smoother. The method will also be demonstrated in a sample application investigating the dynamic behavior of vesicles in general fluid flows.

[1]Support provided by University at Buffalo - SUNY

8:39AM G5.00004 High-order jet-schemes for problems with moving interfaces
, JEAN-CHRISTOPHE NAVE, McGill University, BENJAMIN SEIBOLD, Temple University, RUBEN ROSALES, MIT — Gradient-augmented and more generally jet-schemes belong to a new class of optimally-local and high order methods. In this talk I will present their general formalism and show several applications to level set interface tracking problems. Also, I will try to highlight the ubiquity of the approach and the simplicity of implementation.

8:52AM G5.00005 Computation of droplet interactions in a turbulent flow[1]
, CLAUDIO TORRES, LOUIS ROSSI, University of Delaware — To simulate tropical mesoscale clouds, we compute flows induced by interacting droplets in a turbulent background flow. The calculation of interactions between droplets, modeled as Stokeslets, is the most computationally expensive component in the full cloud simulation. In practice, we have found GMRes to be a superior algorithm for solving this system relative to block-Jacobi and similar schemes, and we report on our attempt to analyze the suitability of this algorithm in general circumstances. Our stability study reveals that a disperse cloud of droplets GMRes will converge robustly. However, when a large number of droplets ($\sim$40) cluster, the condition number grows and GMRes may stagnate. This change in behavior is related to the distribution of the eigenvalues of the matrix. For a uniform disperse set of droplets, we found that the eigenvalues can be enclosed by an ellipse around 1, with some eigenvalues having a small imaginary part. However, for a clustered set of droplets some eigenvalues start to approach zero, indicative of poor GMRes performance. Finally, we will report on our attempts to develop an effective preconditioner to accelerate these calculations on peta-scale computing machines.

[1]NSF OCI 0904934

9:05AM G5.00006 Net Flow in Oscillating Drops activated by Time-periodic Wettability Change
, JUNG MIN OH, Physics of Complex Fluids Group, TNW, University of Twente, 7500AE Enschede, The Netherlands, DOMINIQUE LEGENDRE, Université de Toulouse, INPT, UPS, Institut de Mécanique des Fluides de Toulouse (IMFT), Allée Camille Soula, F-31400 Toulouse, France, FRIEDER MUGELE, Physics of Complex Fluids Group, TNW, University of Twente, 7500AE Enschede, The Netherlands — We use numerical (volume of fluid) simulations to study the net flow in sessile drops driven to oscillate at variable frequency by time-periodic variations of the wettability. Superimposed onto an oscillatory component, we find a net flow oriented upward along the surface from the contact line towards the apex and downward along the symmetry axis of the drop. Analyzing the trajectories of Lagrangian tracer particles, we find that the velocity of the mean flow vanishes at high and low drive frequencies with a broad maximum in between in agreement with recent experimental observations [Mugele et al. Lab Chip 2011]. Superimposed onto this general trend sharp maxima appear at frequencies corresponding to even shape modes of the drops, with an absolute maximum at the resonance frequency of the P_6 mode. The overall trend follows the behavior expected from a Stokes drift model.

9:18AM G5.00007 Sharp Interface LES of Breaking Waves in Orthogonal Curvilinear Coordinates[1] , ZHAOYUAN WANG, JUNGSOO SUH, JIANMING YANG, FREDERICK STERN, University of Iowa — A sharp interface large-eddy simulation (LES) methodology in orthogonal curvilinear coordinates for the breaking waves produced by a body at high Reynolds number is presented. Both gas and liquid phases are considered for the strong interactions between two phases, such as spray dispersion and bubble entrainment. The level-set based ghost fluid method is adopted for sharp interface treatment and a volume-of-fluid method in orthogonal curvilinear coordinates is coupled with the level set method for enhanced interface tracking properties. A Lagrangian dynamic Smagorinsky subgrid-scale model is used for the spatially filtered turbulence closure. Several small-scale cases, bubbles and droplets, are calculated and compared with reference data to validate the sharp interface method on orthogonal curvilinear grids. Wave breaking over a submerged bump and around a wedge-shaped bow is simulated to demonstrate the capabilities of the solver.

[1]This study was supported by ONR.

9:31AM G5.00008 An iterative two-fluid pressure solver based on the immersed interface method[1] , SHENG XU, Southern Methodist University — An iterative solver based on the immersed interface method is proposed to solve the pressure in a two-fluid flow on a Cartesian grid with second-order accuracy in the infinity norm. The iteration is constructed by introducing an unsteady term in the pressure Poisson equation. In each iteration step, a Helmholtz equation is solved on the Cartesian grid using FFT. The combination of the iteration and the immersed interface method enables the solver to handle various jump conditions across a two-fluid interface. This solver can also be used to solve Poisson equations on irregular domains.

[1]This work is supported by the NSF grant DMS 0915237.

9:44AM G5.00009 Direct Numerical Simulation of Density Currents over Rough Walls , KIRAN BHAGANAGAR, XIAOFENG LIU, University of Texas, San Antonio — Direct numerical simulation solver has been developed to simulate density currents over rough walls. The Navier Stokes equations are solved using 4^{th} order vertical velocity equation and a 2^{nd} order vertical vorticity equation. The roughness is introduced using an elegant immersed boundary method (IBM). Spatial discretization is performed using a high order compact finite differences scheme. The time integration is performed using semi-implicit scheme consisting of an explicit 4^{th} order low-storage Runge Kutta scheme for the nonlinear terms and a Crank-Nicolson scheme for the viscous terms. In this talk we will present the fundamental differences in the entrainment due to the presence of 2D and 3D roughness, and we will present an approach to characterize entrainment in terms of the roughness number and Grashoff's number.

9:57AM G5.00010 The Interaction Between Non-Equilibrium Turbulence and a Thermally Ablative Surface[1] , RYAN CROCKER, YVES DUBIEF, University of Vermont — This main interest of this work concerns the interplay between an erosive flow and erodible surface when the erosive process is low enthalpy ablation. Such systems are know to create coherent ablation patters and cascading mass loss rates due the dynamic interactions between the changing topography of the surface and the near wall physics of turbulent flow. These interactions demonstrate a strong coupling between the mass, momentum, and heat transfers in both time and space. In an attempt to elucidate these couplings and their interconnection a direct numerical algorithm has been developed to simulate ablative turbulent flows. The large jump in physical parameters across the interface is handled with the ghost fluid technique and the surface is tracked with level set methods advected and reinitialized with a fully conservative fast marching method. A mass conserving, cut cell, immersed boundary method is employed to simulate the interaction between the momentum equations and the ablating boundary on a non-conforming mesh. Known solutions to conjugate heat transfer, melting and solidification (both Stefan and Neumann) are used to validate the procedure.

[1]Vermont Advanced Computing Center, NSF-CBET, and NASA Epscore

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G6 Turbulence Theory II 309

8:00AM G6.00001 Dependence of decaying homogeneous turbulence on initial/boundary conditions , PEDRO VALENTE, CHRISTOS VASSILICOS, Imperial College London, TURBULENCE, MIXING AND FLOW CONTROL TEAM[1] — The von Kármán-Howarth equation implies an infinity of invariants corresponding to an infinity of different asymptotic behaviours of the double and triple velocity correlation functions at infinite separations. Given an asymptotic behaviour at infinity for which the Birkhoff-Saffman invariant is not infinite, there are either none, or only one or only two finite invariants. If there are two, one of them is the Loitsyansky invariant and the decay of large eddies cannot be self-similar. We examine the consequences of this infinity of invariants on turbulence decay. We then analyse recent wind tunnel data by Krogstad which show that the far-downstream decay of approximately homogeneous isotropic turbulent flows are, invariably, clearly different from Saffman turbulence; and that very clearly marked differences exist between the far downstream turbulence behaviours generated by conventional grids and multiscale cross grids. (See Phys. Lett. A375, 1010-1013 (2011) and http://arxiv.org/pdf/1106.0603)

[1]www.imperial.ac.uk/tmfc

8:13AM G6.00002 Direct Numerical Simulations of Homogeneous Turbulent Shear Flow with Initial Mean Helicity , FRANK JACOBITZ, University of San Diego, KAI SCHNEIDER, Universite de Provence, Marseille, WOUTER BOS, Ecole Centrale de Lyon, MARIE FARGE, Ecole Normale Superieure — Direct numerical simulations of homogeneous turbulent shear flows are performed in order to investigate the impact of mean helicity imposed on the isotropic initial conditions. As the flows are advanced in time, exponential growth of the turbulent kinetic energy is found after flow anisotropy has developed. The mean helicity, however, is observed to decay due to the symmetry properties of the flow. Distributions of helicity are observed to be skewed according to its initial value. A wavelet-based scale-dependent analysis shows that this skewness is largest for large scales of the turbulent motion and decreases for smaller scales. In addition, a trend to two-dimensionalization for large scales of motion and a preference for helical motion at small scales is found. Joint probability distribution functions show a strong correlation of the signs of helicity and super-helicity for all cases, including Gaussian random fields. This correlation supports the conjecture that super-helicity dissipates helicity.

8:26AM G6.00003 Determination of the Turbulent Decay Exponent , J. BLAIR PEROT, CHRIS ZUSI, University of Massachusetts, Amherst — All theories concerning the decay of isotropic turbulence agree that the turbulent kinetic energy has a power law dependence on time. However, there is significant disagreement about what the value of the exponent should be for this power law. The primary theories, proposed by researchers such as Batchelor, Townsend, and Kolmogorov, have the decay exponent at values of 1, 6/5, 10/7, 3/2, 2, and 5/2. The debate over the decay exponent has remained unresolved for many decades because the decay exponent is an extremely sensitive quantity. Experiments have decay times which are too short to be able to accurately differentiate between the various theoretical possibilities, and all prior numerical simulations of decaying turbulence impose the decay rate *a priori* via the choice of initial conditions. In this work, direct numerical simulation is used to achieve very long decay times, and the initial turbulence is generated by the Navier-Stokes equations and is not imposed. The initial turbulence is created by the stirring action of the flow past 768 small randomly placed cubes. Stirring occurs at $1/30^{th}$ of the simulation domain size so that the low wavenumber and large scale behavior of the turbulent spectrum which dictates the decay rate is generated by the fluid and is not imposed. It is shown that in all 16 simulations the decay exponent closely matches the theoretical predictions of Saffman at both high and low Reynolds numbers. Perot, AIP Advances **1**, 022104 (2011).

8:39AM G6.00004 On the local nature of the pressure Hessian in fluid turbulence , LAURENT CHEVILLARD, EMMANUEL LEVEQUE, FRANCESCO TADDIA, Laboratoire de Physique, ENS Lyon, France, CHARLES MENEVEAU, HUIDAN YU, Johns Hopkins University, USA, CARLOS ROSALES, Technical University Federico Santa Maria, Chile — The Lagrangian dynamics of the velocity gradient tensor A in isotropic and homogeneous turbulence depend on the joint action of the self-streching term and the pressure Hessian. Existing closures for pressure effects in terms of A are unable to reproduce one important statistical role played by the anisotropic part of the pressure Hessian, namely the redistribution of the probabilities towards enstrophy production dominated regions. As a step towards elucidating the required properties of closures, we study several synthetic velocity fields and how well they reproduce anisotropic pressure effects. It is found that synthetic (i) Gaussian, (ii) Multifractal and (iii) Minimal Turnover Lagrangian Map (MTLM) incompressible velocity fields reproduce many features of real pressure fields that are obtained from numerical simulations of the Navier Stokes equations, including the redistribution towards enstrophy-production regions. The synthetic fields include both spatially local, and nonlocal, anisotropic pressure effects. However, we show that the local effects appear to be the most important ones: by assuming that the pressure Hessian is local in space, an expression in terms of the Hessian of the second invariant Q of the velocity gradient tensor can be obtained. This term is found to be well correlated with the true pressure Hessian both in terms of eigenvalue magnitudes and eigenvector alignments.

8:52AM G6.00005 The intriguing mechanism of drag reduction by dilute polymer solutions , RAYHANEH AKHAVAN, DONG-HYUN LEE, University of Michigan — The mechanism of drag reduction by dilute polymer solutions is investigated using results from DNS in turbulent channel flow. Drag reductions of up to 56%, corresponding to Virk's MDR, are achieved through a most intriguing, energetically insignificant, yet dynamically significant, mechanism. The cornerstone of this mechanism is a redirection of a small fraction, of no more than 5% on a volume-averaged basis, of the turbulence kinetic energy (TKE) into the elastic energy of the polymer at select turbulent scales. This redirection of energy leads to a decrease in the fluctuating strain-rate at the affected scales, which, in turn, results in a drop in the pressure-strain correlation at these and neighboring scales. The drop in the pressure-strain correlation inhibits the transfer of TKE from the streamwise to cross-stream directions, resulting in a highly anisotropic state at the affected scales. This anisotropy inhibits the normal cascade of TKE to smaller scales. Thus the minute extraction of energy by the polymer initiates a "self-amplifying" sequence of events in which turbulence loses its three-dimensionality and the turbulence energy cascade is inhibited. The magnitude of drag reduction is determined by the range of affected scales, which is a function of We_τ and polymer concentration. For maximum drag reduction, all large scales throughout the channel need to experience the minimal initial extraction of TKE by the polymer.

9:05AM G6.00006 Skin-Friction Drag Reduction in Turbulent Channel Flow by a Scale-Dependent Molecular Viscosity , DONG-HYUN LEE, RAYHANEH AKHAVAN[1], University of Michigan — In prior work, we have proposed that the primary mechanism of drag reduction by dilute polymer solutions is the polymer's extraction of a minute amount of turbulence kinetic energy from the large turbulent scales. Here, we mimic this mechanism by performing DNS with a scale-dependent molecular viscosity in turbulent channel flow. Simulations were performed in channels of size $10h \times 5h \times 2h$ and $40h \times 10h \times 2h$ at a base Reynolds number of $Re_\tau \sim 230$. Drag reductions of 50% and higher were observed when the molecular viscosity was artificially raised from ν_s to $(3-4)\nu_s$ in a band of large-scale wavenumbers corresponding to $0.01 < \sqrt{k_x^2 + k_y^2}/k_d < 0.1$. Many characteristics of drag reduction by dilute polymer solutions were reproduced by the scale-dependent molecular viscosity, including strong anisotropy in the turbulence structure, interruption of the turbulent energy cascade, a pileup of turbulence kinetic energy at the large scales in the streamwise component of the fluctuating velocity, and a shift of the peak of turbulence production away from the wall. These results open up new possibilities for devising novel turbulent skin-friction drag reduction strategies in wall flows.

[1]Supported by The Martin R. Prince Foundation & Teragrid Allocation CTS070070

9:18AM G6.00007 Energy Dissipation Rate in Stably Stratified Turbulence , SABA ALMALKIE, STEPHEN DE BRUYN KOPS, University of Massachusetts, Amherst — One of the key issues in modeling complex flows is the characteristics of small scale turbulence under the influence of large scale anisotropies. We study turbulence dynamics under stabilizing effect of stratification using direct numerical solutions of horizontally homogeneous, vertically stratified turbulence. The simulations use up to $4096 \times 4096 \times 2048$ grid points to resolve the dissipation scales. The Froude number ranges from 0.125 to 1 and the buoyancy Reynolds number from 9 to 219. The small scale turbulence is characterized in terms of one and two-point statistics of the local and locally averaged energy dissipation rate. Intermittency of the energy dissipation rate and its scale dependency gives us insight into the scaling characteristics of the stratified flows and deviation of the scaling laws from those of the isotropic turbulence as a function of Froude number. The effects of stratification on the intermittency of energy dissipation rate and the scales of turbulent bursts are discussed.

9:31AM G6.00008 Strong Thin Shear Layers in High Reynolds Number Turbulence , TAKASHI ISHIHARA, Nagoya University, JST CREST, JULIAN C.R. HUNT, University College London, YUKIO KANEDA, Nagoya University — Data analysis of high resolution DNS of isotropic turbulence with the Taylor scale Reynolds number $R_\lambda = 1131$ shows that there are strong thin shear layers of thickness of the order of Taylor microscale, consisting of a cluster of strong vortex tubes. There are big velocity jumps of the order of the rms of the fluctuation velocity not only across the layers, but also over a small distance of order 10η ($\eta =$the Kolmogorov length scale), due to the strong vortex tubes within the layers. On average there is a balance between the energy dissipation and the energy transfer within the layers where the most intense vortices and dissipation occur. The large fluctuation of the energy transfer implies that not only downscale energy transfer but also upscale energy transfer is very active at the regions adjacent the layer. The DNS data show that the layer as well as the surrounding is moving and there is a net contraction flow on opposite sides of the layer; the interface of the layer is sharper at the side where stronger contraction occurs. The DNS data are consistent with a picture that the downscale and the upscale energy transfers correspond respectively to small eddies being distorted inhomogeneously by larger eddies, as they impinge onto and separate from the thin shear layers.

9:44AM G6.00009 Lagrangian statistics in compressible isotropic homogeneous turbulence[1] , YANTAO YANG, JIANCHUN WANG, YIPENG SHI, SHIYI CHEN, Center for Applied Physics and Technology and SKLTCS, College of Engineering, Peking University, Beijing, China — In this work we conducted the Direct Numerical Simulation (DNS) of a forced compressible isotropic homogeneous turbulence and investigated the flow statistics from the Lagrangian point of view, namely the statistics is computed following the passive tracers trajectories. The numerical method combined the Eulerian field solver which was developed by Wang et al. (2010, *J. Comp. Phys.*, **229**, 5257-5279), and a Lagrangian module for tracking the tracers and recording the data. The Lagrangian probability density functions (p.d.f.'s) have then been calculated for both kinetic and thermodynamic quantities. In order to isolate the shearing part from the compressing part of the flow, we employed the Helmholtz decomposition to decompose the flow field (mainly the velocity field) into the solenoidal and compressive parts. The solenoidal part was compared with the incompressible case, while the compressibility effect showed up in the compressive part. The Lagrangian structure functions and cross-correlation between various quantities will also be discussed.

[1]This work was supported in part by the China's Turbulence Program under Grant No.2009CB724101.

9:57AM G6.00010 Scaling and statistics in three-dimensional compressible turbulence , JIANCHUN WANG, XIANTU HE, YIPENG SHI, SKLTCS and CAPT, College of Engineering, Peking University, China, LIAN-PING WANG, Department of Mechanical Engineering, University of Delaware, Newark, DE 19716, USA, ZUOLI XIAO, SHIYI CHEN, SKLTCS and CAPT, College of Engineering, Peking University, China — The scaling and statistical properties of three-dimensional (3D) compressible turbulence are studied using high resolution numerical simulation. The two-point statistics of solenoidal component of velocity field are found to be no significant difference from those in the incompressible turbulence, while scaling exponents for compressive component of velocity structure functions are saturated at high orders, similar to one-dimensional (1D) Burgers turbulence. Both numerical results and theoretical analysis based on the probability density function (PDF) equation reveal that the power law exponent in the PDF of 3D velocity divergence is different from that in the Burgers turbulence. The effect of various terms in the dynamic equation on PDF will also be presented.

Monday, November 21, 2011 8:00AM - 10:10AM – Session G7 Turbulent Boundary Layers IV 310

8:00AM G7.00001 Artificial large-scale-motion perturbation of a turbulent boundary layer , I. JACOBI, B.J. MCKEON, Caltech — A zero-pressure-gradient flat-plate boundary layer is perturbed dynamically with a spatially-impulsive strip of two-dimensional roughness elements (which alternate with a flush-surface condition periodically in time). The perturbation knocks the boundary layer out of equilibrium and artificially introduces a very-large-scale, periodic structure into the flow. Large-scale- motions in the turbulent boundary layer have recently been understood as a significant source of both turbulent kinetic energy and Reynolds stress, and in addition, have been shown to take part in an apparent amplitude modulation of smaller scales in the flow. The properties of this artificially generated large- scale-motion are studied with particular emphasis on the phase relationship between it and smaller scale structures. The behavior of the artificially-introduced large-scale-motion is also compared with the natural large-scales of the unperturbed turbulent boundary layer, and the interaction between the artificial and natural large-scales is explored. This study is supported by the Air Force Office of Scientific Research under grant #FA9550-08-1-0049 (program manager John Schmisseur).

8:13AM G7.00002 What happens to wall-bounded flows when the mean velocity profile is artificially prescribed?[1] , JAVIER JIMENEZ, U. Politécnica Madrid and CTR Stanford, FLORIAN TUERKE, UPM and Technische U. Berlin — Direct numerical simulations of turbulent channels with artificially prescribed mean velocity profiles, both natural and purposely unnatural, are used to study the dynamics of the energy-containing turbulent fluctuations. It is found that turbulence develops correctly in natural profiles, but that even slightly incorrect mean velocity gradients modify the intensities and Reynolds stresses substantially. The extra energy created by a locally stronger imposed shear resides in structures with essentially correct dimensionless ratios, but which are out of energy equilibrium. In profiles with sharp shear jumps in the logarithmic layer, the relaxation rates away from the discontinuity are different for the kinetic energy, the tangential Reynolds stress, and the dissipation, but they are all consistent with the turbulent advection of eddies with relaxation times of the order of the local eddy turnover. Correspondingly, the energy imbalance is compensated by turbulent diffusion. Interestingly, only eddies above a certain size are created by the discontinuity. The smallest ones do not immediately attach to the wall, but those about twice as large do.

[1]Funded by CICYT and ERC.

8:26AM G7.00003 Comparison of the one- and two-point second-order statistics for zero-pressure-gradient turbulent boundary layers and channels at $Re_\tau \approx 1000 - 2000$[1] , JUAN A. SILLERO, AYSE G. GUNGOR, JAVIER JIMÉNEZ, U. Politecnica Madrid, ROBERT D. MOSER, U. Texas at Austin — One and two-points statistics are presented from a new direct simulation of the zero-pressure-gradient turbulent boundary layer in the range $Re_\theta = 2780 - 6680$, matching channel flow simulations at $Re_\tau \approx 1000-2000$. The integral parameters, mean velocities, Reynolds stresses and pressure fluctuations of the boundary layer closely agree with the numerical and experimental data sets available in the literature, but show clear differences with channels, when expressed in wall units at the same y/δ. Those differences seem to increase, rather than decrease, with the Reynolds number. Spectra and spatial correlations $C_{\xi\xi}(x;x';y;y';k_z)$ are also investigated, and include a range of scales of the energy-containing structures larger than an order of magnitude.

[1]Funded by CICYT, INCITE and ERC.

8:39AM G7.00004 Effect of the outer laminar flow on zero-pressure-gradient turbulent boundary layers[1] , YOSHINORI MIZUNO, CALLUM ATKINSON, JULIO SORIA — This study examines the effect of the invasion of the laminar flow on zero-pressure-gradient turbulent boundary layers by using direct numerical simulations, comparing with channel flows for comparable Reynolds numbers around $\delta^+ = 600$, where δ is the boundary layer thickness and $^+$ stands for normalization by a wall-unit. The turbulent/non-turbulent boundary is captured in an appropriate way, and the transitional layer, often called viscous (or laminar) superlayer, is defined. The distribution of the position of the boundary is represented well by the Gaussian distribution function of the wall-distance, and the mean and standard deviation are found to be approximately 0.9δ and 0.15δ, respectively. A clear gap in turbulence intensities is observed within this layer, and enstrophy and dissipation rate are locally enhanced by the strong shear in the turbulence-side of this layer. The thickness of the viscous superlayer is found to be in the order of the Kolmogorov's length as found in previous works, but it is also found that it becomes thinner as moving away from the wall. The enhancement of enstrophy and dissipation rate is more significant at higher positions.

[1]This research is supported by the Discovery Grant from the Australian Research Council and the National Computational Infrastructure (NCI).

8:52AM G7.00005 Development of a turbulent boundary layer beneath finite-amplitude continuous freestream turbulence

, XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, CTR, Stanford University — Following the earlier work of Wu & Moin (JFM 2009, PoF 2010) and Wu (JFM 2010), here we will present our third, most recent, direct numerical simulation of the incompressible, zero-pressure-gradient flat-plate boundary layer. Heat transfer between the constant-temperature plate and the free-stream is also simulated with unit molecular Prandtl number. The freestream of the present boundary layer has continuous isotropic turbulence whose inlet strength is 3% of the mean velocity. Its decay characteristics agree with existing water channel experiments. Despite the finite-level freestream perturbation, the boundary layer is clean in the sense that the deviation of skin-friction from Blasius prior to breakdown is less than 1%. Both the statistics and structures from this simulation will be compared with our previous DNS studies using periodically fed patches of isotropic turbulence. The associated bypass transition process will also be evaluated.

9:05AM G7.00006 Direct numerical simulation of K-type and H-type transitions to turbulence in a low Mach number flat plate boundary layer[1]

, TARANEH SAYADI, CURTIS HAMMAN, PARVIZ MOIN, Center for Turbulence Research (CTR), Stanford University — Transition to turbulence via spatially evolving secondary instabilities in compressible, zero-pressure-gradient flat plate boundary layers is numerically simulated for both the Klebanoff K-type and Herbert H-type disturbances. The objective of this work is to evaluate the universality of the breakdown process between different routes through transition in wall-bounded shear flows. Each localized linear disturbance is amplified through weak non-linear instability that grows into lambda-vortices and then hairpin-shaped eddies with harmonic wavelength, which become less organized in the late-transitional regime once a fully populated spanwise turbulent energy spectrum is established. For the H-type transition, the computational domain extends from $Re_x = 10^5$, where laminar blowing and suction excites the most unstable fundamental and a pair of oblique waves, to fully turbulent stage at $Re_x = 10.6 \times 10^5$. The computational domain for the K-type transition extends to $Re_x = 9.6 \times 10^5$. The computational algorithm employs fourth-order central differences with non-reflective numerical sponges along the external boundaries. For each case, the Mach number is 0.2.

[1]Supported by the PSAAP program of DoE, ANL and LLNL.

9:18AM G7.00007 DNS of laminar/turbulent boundary layer transition induced by solid obstacles[1]

, PAOLO ORLANDI, MATTEO BERNARDINI, SERGIO PIROZZOLI, Università di Roma "La Sapienza" — Direct numerical simulation is used to investigate how boundary layer transition is affected by the shape and size of an isolated obstacle whose size is of the order of the boundary layer thickness. The Navier-Stokes equations are discretized by means of an energy-conserving second-order staggered finite-difference method, and the geometrical complexity associated with the obstacle is handled through the immersed-boundary technique. A series of simulations have been performed by varying: i) the obstacle shape (cylinders and prisms with rectangular and triangular base); ii) the roughness height (as a fraction of the boundary layer thickness); iii) the width of the obstacle; iv) the Reynolds number of the incoming boundary layer. We have monitored the vorticity dynamics of the structures which are shed past the obstacle, and observed the concurrence of two mechanisms which promote transition to turbulence, namely the unsteady shear layer separation at the top edge of the obstacle, and the regeneration of quasi-streamwise vortices at the sides of the obstacle. The validity of semi-empirical transition criteria based on a suitably defined roughness Reynolds number will also be discussed, and associated with the physical mechanisms responsible for the self-sustainment of the disturbances.

[1]The computational resources to perform the study were provided by CINECA.

9:31AM G7.00008 Turbulent boundary layers: Inflow effects and cross-validation of simulation and experiment

, RAMIS OERLUE, PHILIPP SCHLATTER, Linne FLOW Centre, KTH Mechanics, Royal Institute of Technology, Stockholm, SWEDEN — A recent assessment of available direct numerical simulation (DNS) data from turbulent boundary layer flows [Schlatter & Örlü, J. Fluid Mech. 659, 116 (2010)] showed surprisingly large differences not only in the skin friction coefficient or shape factor, but also in their predictions of mean and fluctuation profiles far into the sublayer. Several DNS of a zero pressure-gradient (ZPG) turbulent boundary layer (TBL) à la Schlatter et al. [Phys. Fluids 21, 051702 (2009)] with physically different inflow conditions and tripping effects were performed. Most of the differences observed when comparing available DNS could thereby be traced back to different initial conditions. It was also found, that if transition is initiated at a low enough Reynolds number (based on the momentum-loss thickness) $Re_\theta <$ 300, all data agree well for both inner and outer layer for $Re_\theta > 2000$; a result that gives a lower limit for meaningful comparisons between numerical and/or wind tunnel experiments. Based on these results a detailed comparison between DNS and experiment of a ZPG TBL flow at $Re_\theta = 2500$ and 4000 is presented. Good agreement is obtained for integral quantities, mean and fluctuating streamwise velocity profiles, but also for the probability distribution and spectral map throughout the boundary layer.

9:44AM G7.00009 Turbulent boundary layers in long computational domains

, PHILIPP SCHLATTER, QIANG LI, RAMIS OERLUE, GEERT BRETHOUWER, ARNE V. JOHANSSON, P. HENRIK ALFREDSSON, DAN S. HENNINGSON, KTH Mechanics — A new series of numerical simulations of spatially evolving turbulent boundary layers is discussed. The very long computational domain starts at a low $Re_\theta = 180$, where laminar-turbulent transition is initiated, reaching up to the (computationally very) high $Re_\theta = 8500$. In the domain, the boundary layer develops naturally from the tripping location to the higher Reynolds numbers without any re-injection or recycling procedures. In consequence, this computational setup allows us to study, e.g., the mean flow development and the scaling behavior of the fluctuating energy free from pseudo-periodic effects. However, such domains require a large number of grid points; in the present case up to 10 billion for running well-resolved large-eddy simulation. The present results show excellent agreement with wind-tunnel experiments at similar Re and previous (lower-Re) simulations (both direct and large- eddy simulations). The mean velocity profiles closely follow the correlation proposed by Monkewitz et al. (2007), just about reaching the plateau in the log-law diagnostic function. In a second part, three-dimensional visualizations of the evolving turbulent boundary layer are discussed with special focus on the persistence of transitional flow structures towards higher Reynolds numbers, having a highly unordered appearance.

9:57AM G7.00010 Turbulence statistics in turbulent spots in a transitional boundary layer subject to free-stream turbulence

, BRENDAN REHILL, Stokes Institute, University of Limerick, Limerick, Ireland, ED J. WALSH COLLABORATION[1], PHILIPP SCHLATTER, LUCA BRANDT COLLABORATION[2], TAMER A. ZAKI COLLABORATION[3], DONALD M. MCELIGOT COLLABORATION[4] — Within the boundary layer transition region turbulent spots emerge and grow to form the fully-turbulent boundary layer. This paper examines the turbulent statistics within turbulent spots in a transitional boundary layer subject to free-stream turbulence intensity of 4.7%. Conditionally sampled DNS results, where the laminar and turbulent contributions to the transition region are separated, are used to obtain the relevant statistics. Conditional sampling of the data provides some improvement over the more classical time-space-averaged data reduction techniques, through providing more insight into the true turbulent statistics within turbulent spots. The statistics are compared to the lowest fully-turbulent DNS available in the literature to identify how the turbulent spots develop and form the fully-turbulent state.

[1]Stokes Institute, University of Limerick, Limerick, Ireland
[2]Linne Flow Centre, KTH Mechanics, Stockholm, Sweden
[3]Dept. Mech. Engr., Imperial College London, London, UK
[4]Mech. Engr. Dept., University of Idaho, Idaho Falls, Idaho, USA

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G8 Low Reynolds Number Swimming II 311

8:00AM G8.00001 Stability of an array of bottom-heavy, upswimming, spherical squirmers , TIMOTHY PEDLEY, DOUGLAS BRUMLEY, University of Cambridge — Numerical simulations by Ishikawa & Pedley (Phys Rev Lett. 100:088103,2008; IP) of identical, bottom-heavy, spherical squirmers swimming in a vertical planar monolayer reveal a configuration in which many equally-spaced spheres swim upwards in a hexagonal array. This occurs for sufficiently large values of the bottom-heaviness parameter G. Here we perform an instability analysis in which the forces and torques on individual spheres are calculated using lubrication theory for the gaps between neighbouring squirmers. The results show that stability is impossible in the absence of the repulsive force which IP introduced to save computer time whenever two squirmers were less than 0.00001 radii apart. When the force is present, there is a critical value of G above which stability is assured. An unexpected finding is the existence of another equilibrium configuration in which the spheres' orientations are not vertical and columns of spheres are not equally spaced; in this case, however, perturbation leads to limit-cycle oscillations, not to static stability. The above results are qualitatively unaffected if the monolayer is confined between vertical walls.

8:13AM G8.00002 Investigations of the Response of Swimming Paramecia to Variations in their Apparent Weight[1] , JAMES VALLES, ILYONG JUNG, Brown University, KARINE GUEVORKIAN, Curie Institute, HARRY MICKALIDE, MICHAEL WAGMAN, Brown University — There is a set of micro-organisms that are small enough that they swim at low Reynolds number and large enough that gravity exerts an influence on their behavior Many protists, like paramecia, for example, exhibit negative gravi-taxis by orienting their swimming upward and negative gravi-kinesis by increasing their propulsion when swimming against their apparent weight. It is not clear whether these responses to a very weak force (about 100 pN) are active or passive. We have developed a technique, Magnetic Force Buoyancy Variation, which enables us to vary the apparent weight of the swimmers in situ. We will describe experiments on paramecia conducted at the National High Magnetic Field Laboratory. In particular, we will describe how increasing the apparent weight induces paramecia to accumulate at upper surfaces. A simple force model suggests that this accumulation is a passive response.

[1]Supported by NSF-PHY0750360 and a grant to the NHMFL, NSF DMR-0654118.

8:26AM G8.00003 Locomotion of a Reciprocal Swimmer by Fluid Elasticity , NATHAN C. KEIM, MIKE GARCIA, PAULO E. ARRATIA, Dept. of Mechanical Engineering and Applied Mechanics, University of Pennsylvania — When fluid response is entirely viscous, a swimmer performing a reciprocal motion achieves no net displacement. This form of time-reversal symmetry is commonly broken by a non-reciprocal swimming stroke, but it may also break down if the fluidic environment has a nonlinear viscoelastic response, as found in many natural media such as mucus. In this talk, we present experiments on a rigid dimer that is "wiggled" in a reciprocal motion by a magnetic field, in the vicinity of a wall. When the dimer is immersed in a viscoelastic fluid, its motion produces a net translation. Surprisingly, the dimer can swim in a direction that is primarily parallel to the wall. No net translation is seen in a viscous Newtonian fluid under the same conditions. We report the effect's dependence on Deborah number, swimming stroke, and geometric parameters. The underlying mechanism is examined with particle tracking measurements.

8:39AM G8.00004 Optimal ciliary beating patterns , ANDREJ VILFAN, NATAN OSTERMAN, J. Stefan Institute, Ljubljana, Slovenia — We introduce a measure for energetic efficiency of single or collective biological cilia. We define the efficiency of a single cilium as Q^2/P, where Q is the volume flow rate of the pumped fluid and P is the dissipated power. For ciliary arrays, we define it as $(\rho Q)^2/(\rho P)$, with ρ denoting the surface density of cilia. We then numerically determine the optimal beating patterns according to this criterion. For a single cilium optimization leads to curly, somewhat counterintuitive patterns. But when looking at a densely ciliated surface, the optimal patterns become remarkably similar to what is observed in microorganisms like *Paramecium*. The optimal beating pattern then consists of a fast effective stroke and a slow sweeping recovery stroke. Metachronal waves lead to a significantly higher efficiency than synchronous beating. Efficiency also increases with an increasing density of cilia up to the point where crowding becomes a problem. We finally relate the pumping efficiency of cilia to the swimming efficiency of a spherical microorganism and show that the experimentally estimated efficiency of *Paramecium* is surprisingly close to the theoretically possible optimum.

[1] N. Osterman and A. Vilfan, Finding the ciliary beating pattern with optimal efficiency, Proc. Natl. Acad. Sci. USA, in press (2011)

8:52AM G8.00005 Optimal feeding vs. optimal swimming of model ciliates , SEBASTIEN MICHELIN, Ecole Polytechnique - LadHyX, ERIC LAUGA, UCSD-MAE — To swim at low Reynolds numbers, micro-organisms create flow fields that modify the transport of nutrients around them, thereby impacting their feeding rate. When the nutrient is a passive scalar, the feeding rate of a given micro-swimmer greatly varies with the Péclet number (Pe) a relative measure of advection and diffusion in the nutrient transport, that strongly depends on the nutrient species considered. Using an axisymmetric envelope model for ciliary locomotion and adjoint-based optimization, we determine the swimming (or possibly non-swimming) strokes maximizing the nutrient uptake by the micro-swimmer for a given energy cost. We show that, unlike the feeding rate, this optimal feeding stroke is essentially independent of the Péclet number (and, therefore, of the nutrient considered) and is identical to the stroke with maximum swimming efficiency.

9:05AM G8.00006 Sea Butterfly Swimming: Time-resolved Tomographic PIV measurements , DAVID MURPHY, Civil and Environmental Engineering, Georgia Tech, LINGXIAO ZHENG, RAJAT MITTAL, Mechanical Engineering, Johns Hopkins University, DONALD WEBSTER, Civil and Environmental Engineering, JEANNETTE YEN, Biology, Georgia Tech — The planktonic sea butterfly *Limacina helicina* swims by flapping its flexible, wing-like parapodia. The appendage stroke kinematics of this shell-bearing pteropod are three-dimensional and likely contain elements of both drag-based (rowing) and lift-based (flapping) propulsion. Unsteady lift-generating mechanisms such as clap-and-fling may also be present. Upstroke and downstroke motions both propel the animal upward and roll it forwards and backwards, resulting in a sawtooth trajectory. We present time-resolved, tomographic PIV measurements of flow generated by free-swimming pteropods (*Limacina helicina*) moving upwards with average swimming speeds of 5 – 17 mm/s. The pteropods beat their appendages with a stroke frequency of 4 – 5 Hz. With a size range of 1 – 2 mm, the animals filmed in this study operate in a viscous environment with a Reynolds number of 5 to 20. The volumetric flow measurements provide insight into the three dimensional nature of the flow and into the relative importance of drag- and lift-based propulsion at this low Reynolds number. Preliminary results from Navier-Stokes simulations of the flow associated with the swimming of this organism will also be presented.

9:18AM G8.00007 Effect of phase delay on the pumping efficiency of a multi-plate gill array[1] , MARY LARSON, KEN KIGER, Univ. of Maryland — In nature, pumping by oscillating appendage arrays (used for respiration, feeding or locomotion) have been noted to exhibit distinct patterns of movement depending on their intended function and Reynolds number. One thing that is typically in common, however, is that a phase lag of 60 to 90 degrees between adjacent appendages is used for many low to intermediate Reynolds number conditions (10 to 10000). To understand why this trait is commonly exhibited, a robotic oscillating plate array modeled after a nymphal mayfly was constructed that permitted stroke, pitch and phase lag variation between adjacent plates. Stereoscopic PIV was used to obtain three-dimensional velocity data, allowing computation of the net pumping rate and flow induced dissipation for five cases, focusing on the role of the gill plate interactions and their dependence on the phase lag between adjacent gills. The results indicate that mayfly gills most likely use a phase lag of 90° because it produces the highest net mass flow rate while consuming the least amount of energy. Measurements indicate that this occurs as a balance between excessive dissipation during close-approach events while optimizing favorable hydrodynamic interactions between adjacent plates.

[1]Work supported by NSF under grant CBET0730907.

9:31AM G8.00008 A numerical study on swimming micro-organisms inside a capillary tube[1] , LAILAI ZHU, Linne Flow Centre, KTH Mechanics, ERIC LAUGA, Department of Mechanical and Aerospace Engineering, University of California San Diego, LUCA BRANDT, Linne Flow Centre, KTH Mechanics — The locomotivity of micro-organisms is highly dependent on the surrounding environments such as walls, free surface and neighbouring cells. In our current work, we perform simulations of swimming micro-organisms inside a capillary tube based on boundary element method. We focus on the swimming speed, power consumption and locomotive trajectory of swimming cells for different levels of confinement. For a cell propelling itself by tangential surface deformation, we show that it will swim along a helical trajectory with a specified swimming gait. Such a helical trajectory was observed before by experiments on swimming *Paramecium* inside a capillary tube.

[1]Funding by VR (the Swedish Research Council) and the National Science Foundation (grant CBET-0746285 to E.L.) is gratefully acknowledged. Computer time provided by SNIC (Swedish National Infrastructure for Computing) is also acknowledged.

9:44AM G8.00009 Swimming of Flexible Nanowire Motors , ON SHUN PAK, Department of Mechanical and Aerospace Enginering, University of California San Diego, WEI GAO, JOSEPH WANG, Department of Nanoengineering, University of California San Diego, ERIC LAUGA, Department of Mechanical and Aerospace Enginering, University of California San Diego — In this talk, we report on a new nanowire motor which exploits the flexibility of nanowires for propulsion. The motor is made of nickel and silver nanowires, and it is fabricated using a common template-directed electrodeposition protocol. These readily prepared nanomotors display both high dimensional and dimensionless (in body lengths per revolution) propulsion velocities when compared with natural microorganisms and other artificial propellers. Their propulsion characteristics are studied theoretically using an elastohydrodynamic model which takes into account the elasticity of the nanowire and its hydrodynamic interaction with the fluid medium. The theoretical predictions by an asymptotic analysis for small-amplitude swimming are then compared with experimental measurements and we obtain good agreement. Finally, we demonstrate the operation of these nanomotors in a real biological environment (human serum), emphasizing the robustness of their propulsion performance and their promise for biomedical applications.

9:57AM G8.00010 Electrokinetic self-Propulsion of a Catalytic Nanomotor; a Perturbation Analysis[1] , AMIR NOURHANI, Chem. Eng. Dept., Penn State University, PAUL LAMMERT, Phys. Dept., Penn State University, ALI BORHAN, Chem. Eng. Dept., Penn State University, VINCENT CRESPI, Phys. Dept., Penn State University — The development of bimetallic catalytic nanomotors has been one of many attempts to mimic the behavior of bionanomotors and micro-organisms. Recent experimental analysis has shown that electrokinetic self-propulsion is the dominant mechanism for autonomous motion of these nanomotors in hydrogen peroxide. In this work, we propose a mathematical model for the steady state locomotion of an axisymmetric spherical nanomotor in a symmetric binary electrolyte. The asymmetric production and consumption of hydrogen ions on the surface of the particle is modeled by a general position-dependent flux. The flux of negative ions on the surface is zero. Using perturbation analysis and the method of matched asymptotic expansion, we solve the model for the velocity of the particle to the leading order in Debye length and to the first order in the intensity of hydrogen ions flux. The velocity depends linearly on the interfacial potential of the surface of the particle and the intensity of hydrogen ions flux. It also inversely depends on the viscosity of fluid, the ion concentration in electrolyte and the diffusion coefficient of the hydrogen ions. The predicted behavior is consistent with experimental results and numerical calculations.

[1]The work is funded by MRSEC at Penn State University.

Monday, November 21, 2011 8:00AM - 9:31AM – Session G10 MHD I 313

8:00AM G10.00001 Quantifying the locality/nonlocality of nonlinear interactions in MHD turbulence , J.A. DOMARADZKI, USC, B. TEACA, EPF Lausanne, D. CARATI, ULB 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 (HD) confirmed the theoretical scaling exponent of n=4/3 in the wavenumber ratio k/k_c. We have extended the analysis of DNS data to MHD turbulence. The analysis is performed in spectral space, which is decomposed into a series of shells following a power law for the boundaries. The triadic transfers occurring among these shells are computed and the fluxes and locality functions are recovered by partial summations over the relevant shells. Values of 1/3 and 2/3 are found for the scaling exponents of the four individual MHD energy fluxes corresponding to the four nonlinear terms in MHD equations. However, when a sum of two energy conversion terms, among the kinetic and the magnetic energy, is considered instead of its two individual components, its scaling exponent is 2/3, the same as for the remaining two redistribution terms for the kinetic and the magnetic energy. All scaling exponents are smaller than the value of 4/3 found for HD turbulence, indicating significantly more nonlocal character of nonlinear interactions in MHD turbulence.

8:13AM G10.00002 Nonlocality and the Critical Reynolds Numbers of the Minimum State Magnetohydrodynamic Turbulence[1] , YE ZHOU, Lawrence Livermore National Laboratory, Livermore, California, 94550, USA, SEAN OUGHTON, Department of Mathematics, University of Waikato, Hamilton, New Zealand — Magnetohydrodynamic (MHD) systems can be strongly nonlinear (turbulent) when their kinetic and magnetic Reynolds numbers are high, as is the case in many astrophysical and space plasma flows. Unfortunately these high Reynolds numbers are typically much greater than those currently attainable in numerical simulations of MHD turbulence. A natural question to ask is how can researchers be sure that their simulations have reproduced all of the most influential physics of the flows and magnetic fields? In this talk, a metric is defined to indicate whether the necessary physics of interest has been captured. It is found that current computing resources will typically not be sufficient to achieve this minimum state metric.

[1]This work was performed under the auspices of the Lawrence Livermore National Security, LLC under contract No. DE-AC52-07NA27344 and with support from the University of Waikato Strategic Research Investment Fund.

8:26AM G10.00003 Numerical Study for the MHD Homogeneous Decaying Turbulence with the Uniform Magnetic Field , MASAYOSHI OKAMOTO, Department of the Mechanical Engineering, Shizuoka University — The MHD homogeneous decaying turbulent flows with the uniform magnetic field at three magnetic Prandtl numbers are investigated by means of the direct numerical simulation. The decay of the total energy, which is the sum of the kinetic and magnetic energy, is relaxed due to the additional magnetic field. The anisotropy of the Reynolds and Maxwell normal stresses is not large. However, in the power spectra of the velocity and magnetic fields the small-scale anisotropic property is under the influence of the constant magnetic field. From the viewpoints of the energy spectral budget, the contribution of the energy transformation term related with the mean magnetic field is dominant in comparison with that related with only fluctuating field.

8:39AM G10.00004 Turbulent transport of passive scalar in magnetohydrodynamic channel flow[1] , PRASANTA DEY, OLEG ZIKANOV, University of Michigan - Dearborn — Direct numerical simulations are conducted to analyze transport of a passive scalar in a turbulent flow of an electrically conducting fluid in a channel. Cases of imposed wall-normal, spanwise, and streamwise magnetic field are considered. The magnetic Reynolds and Prandtl numbers are assumed small. The hydrodynamic Reynolds number based on the channel half-width and mean velocity is Re=6000 and the Hartmann number varies from zero to the value slightly below the laminarization threshold. We find that the flow transformation caused by the magnetic field leads to significant changes of the statistical properties of the scalar distribution and of the rate of scalar transport. A particularly important factor is the suppression of turbulent fluctuations of wall-normal velocity in the cases of wall-normal and spanwise magnetic fields.

[1]The work is supported by the NSF through grant CBET 096557.

8:52AM G10.00005 Magnetohydrodynamic duct flow at $Re = 10^5$ and strong magnetic field[1] , OLEG ZIKANOV, University of Michigan - Dearborn, DMITRY KRASNOV, THOMAS BOECK, Ilmenau University of Technology — We present the results of high-resolution DNS of the magnetohydrodynamic flows in a duct of square cross-section. The walls of the duct are electrically insulating and the imposed magnetic field is constant, uniform, and parallel to one set of walls. The simulations are performed at the Reynolds number based on the mean velocity and duct half-width $Re = 10^5$ and the Hartmann number Ha varying from 0 to 400, i.e. in the range of parameters never before addressed in computational analysis. The numerical model is based on a conservative finite-difference scheme and uses the grid consisting of up to 2048×768^2 points. The results show a sequence of flow regimes that appear with increasing Ha: a turbulent flow with suppressed momentum transfer in the sidewall (parallel to the magnetic field) boundary layers, a flow with laminar core and turbulent sidewall layers, a flow with laminar core and quasi-two-dimensional structures in the sidewall layers, and, at Ha=400, a completely laminarized flow.

[1]The work is supported by the DFG through Graduiertenkollegs GRK 1567 and by the NSF through grant CBET 096557.

9:05AM G10.00006 On the accurate simulation of magnetohydrodynamic turbulence and magnetic reconnection , MINPING WAN, University of Delaware, SEAN OUGHTON, University of Waikato, SERGIO SERVIDIO, Universita della Calabria, WILLIAM H. MATTHAEUS, University of Delaware — The issue of adequate spatial resolution in numerical simulations of turbulence is studied in the context of decaying two-dimensional magnetohydrodynamics. By comparing simulations at varying resolution and at varying Reynolds numbers, the familiar criterion that the dissipation scale should be resolved is found to enable accurate computation of the spectrum, but fail for precise determination of higher-order statistical quantities. Examination of two straightforward diagnostics, the maximum of the kurtosis and the scale-dependent kurtosis, enables the development of simple tests for assessing adequacy of spatial resolution. The efficacy of the tests is confirmed by examining the distribution of magnetic reconnection rates in turbulence. Over-sampling the Kolmogorov dissipation scale by a factor of 3 allows accurate computation of the kurtosis, the scale-dependent kurtosis, and the reconnection rates. These tests may provide useful guidance for resolution requirements in many plasma computations involving turbulence and reconnection. [Phys. Plasmas 17, 082308 (2010)]

9:18AM G10.00007 Experimental investigation of free surface fluctuations and vortex dynamics in MHD flow[1] , J. RHOADS, PPPL, J. KUBRICHT, UT Austin, E. EDLUND, P. SLOBODA, E. SPENCE, H. JI, PPPL — A magnetic field imposed on a conducting liquid can significantly alter the dynamics of large and small scale features within the flow. These effects may be critically important in turbulent heat transport for flowing liquid metal walls in a fusion reactor. Experiments have been conducted in the Liquid Metal Experiment (LMX) using a GaInSn eutectic alloy as a working fluid to investigate these effects. These experiments considered free-surface, wide aspect-ratio flow through a channel situated in a strong vertical magnetic field (up to $\mathrm{Ha} \approx 50$, where Ha is the ratio of electromagnetic to viscous forces). By tracking the deflection of the free surface in three locations, correlation analysis in both the down-stream and cross-stream directions gave insight into how the fluctuations were affected by the application of the magnetic field. Additionally, vorticies created in the wake of an obstruction responded dramatically to the applied field. Experimental results examining the damping of surface fluctuations and the characterization of vortices being shed from an obstruction will be presented.

[1]Work supported under contract DE-AC02-09CH11466.

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G11 Granular Flows II 314

8:00AM G11.00001 Still waters: deadzone formation in granular jet impact , JAKE ELLOWITZ, HERVE TURLIER[1], NICHOLAS GUTTENBERG, WENDY W. ZHANG, SIDNEY R. NAGEL, James Franck Institute, Chicago, IL — When a densely packed jet of non-cohesive grains collides with a target, a deadzone of stagnant particles forms within the jet. Analogous deadzones form when a droplet of dense suspension hits a substrate, a process important in coating and inkjet printing. In a different context, the formation of planets depends on successive collisions and aggregation of centimeter sized dust particles. The growth process is only possible if a deadzone is deposited onto the larger particle. Using experiments, discrete particle simulations and continuum modeling, we elucidate factors leading to deadzone formation in the collisions in noncohesive dense granular jet impact. We find that a frictional fluid model quantitatively reproduces the structure and stresses observed in both experiment and simulation. A deadzone does not form when the target is frictionless. In contrast, when the target shape is changed from a flat circular disk to a cone, the size of the deadzone decreases continuously with the cone angle.

[1]Currently at Institut Curie, Paris, France

8:13AM G11.00002 Granular Response to an Impact[1] , LOU KONDIC, New Jersey Institute of Technology , ROBERT BEHRINGER, Duke University, WOLFGANG LOSERT, University of Maryland, COREY O'HERN, Yale University — In this computational study we discuss response of a granular system to an impact of a large intruder. We find that interaction between the granular particles strongly influences the intruder's dynamics, with particularly strong role played by friction. We will also discuss the dynamics of granular particles to an impact and correlate affine and non-affine components of particles' motion to the dynamics of intruder itself.

[1]Supported by DTRA Grant No. 1-10-1-0021 and NSF Grant No DMS-0835611

8:26AM G11.00003 Impact Dynamics in a 3D Granular Bed[1] , KERSTIN NORDSTROM, MATT HARRINGTON, WOLFGANG LOSERT, University of Maryland — Granular impacts have been studied for many years in the scientific community. Force laws have been characterized for different systems including universal scaling relations by Katsuragi and Durian. Despite this, little is known regarding the microscopic origin of these observations. In this work, we study the impact of a projectile onto a bed of 3 mm grains immersed in an index-matched fluid. Using a laser sheet scanning technique, a high speed camera, and particle tracking, we can measure the trajectory of each grain throughout an impact event. We have characterized the nonaffine motion within the system as a function of projectile shape and initial sample preparation. Our preliminary results show significant nonaffine motion near the impactor. These results are compared and contrasted with recent experiments and simulations involving 2D systems.

[1]Supported by the Defense Threat Reduction Agency.

8:39AM G11.00004 ABSTRACT WITHDRAWN —

8:52AM G11.00005 Study of 2D Granular Impact Dynamics with Photoelasic Particles[1] , ABE CLARK, R.P. BEHRINGER, Duke University, LOU KONDIC, New Jersey Institute of Technology — What is the response of a granular material to a high speed impact from a foreign object? Our experiments consist of a large 2D granular system which is subjected to impact by an intruder from above. Using photoelastic discs and a high-speed camera (up to 775,000 fps), we are able to observe the dynamics in this process in a way which has not been done previously. We compare results to previous work on this subject, which primarily consists of macroscopic frictional models. We observe phenomena which are not captured by averaged friction models, such as large fluctuations in the acceleration of the intruder as it moves through the granular material. The high frame rates also allow us to observe rich photoelastic behavior, including propagation of acoustic waves. The photoelastic response beneath the intruder and the measured acceleration are well correlated, suggesting that a grain-scale, stick-slip description is necessary to account for the dissipation of the intruder's kinetic energy, and that much of the energy might be carried away by compressional waves.

[1]This work is supported by DTRA, grant HDTRA1-10-0021.

9:05AM G11.00006 Dynamics in cohesive granular flow[1] , JENNIFER RIESER, WENBIN LI, JU LI, University of Pennsylvania, JERRY GOLLUB, Haverford College, DOUGLAS DURIAN, University of Pennsylvania — Inter-particle interactions often have a dramatic effect on granular flow dynamics. Here, we explore the flow resulting from an applied compressive stress on a granular pillar composed of a single layer of particles. Grain-grain attractions within the pillar are governed by tunable capillary forces induced by an interstitial fluid. Both the applied stress and the grain positions are monitored as a function of time. We determine the probability distributions of particle translational and angular velocities, local packing fraction, and the number of nearest neighbors as a function of time. In addition, instantaneous velocities are used to characterize larger scale spatial structures that emerge in the flow from the collective motion of particles. We see a striking dependence on initial conditions, for instance in the development and in the behavior of slip planes, both of which are found to be qualitatively similar to molecular dynamics simulations.

[1]This work is supported by MRSEC/DMR05-20020.

9:18AM G11.00007 Blowing in the wind: the dynamics of small laboratory-scale dunes , YANG ZHANG, Department of Fluid Engineering, Xian Jiaotong University, WILLEM VAN DE WATER, Department of Applied Physics, Eindhoven University of Technology — Barchan dunes can be found in the desert under steady wind conditions where they translate unaltered in the direction of the wind. 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. The length scales of these dunes, set by saltation of the grains, can be reduced dramatically by oscillating gravity. This makes laboratory experiments possible. We study the fate of tiny dunes sitting on an oscillating table in the turbulent boundary layer of a wind tunnel. Growth or death of these dunes depends on the influx of sand. Paradoxically, more influx can lead to more erosion. We explain this through a continuum model, and link its unknowns to detailed measurements of grain dynamics.

9:31AM G11.00008 Avalanche dynamics of dry or fully immersed granular mass and temporal evolution of failure structure , YUNG-TA HUANG, FULING YANG, Department of Mechanical Engineering, National Taiwan University, Taipei, Taiwan — This work examines the avalanche process of dry and fully immersed granular mass abruptly released down a laboratory flume (of width 15 cm) of smooth bed at fixed inclination experimentally. The granular mass is composed of identical glass spheres (of diameter 1.6 cm) and Newtonian fluids (water or 200 cP glycerol). High-speed imaging technique was applied to capture individual sphere motion from the flume lateral side. The obtained sphere collective motion show distinctive patters in relation with the failure plane which developed during an avalanche. We also estimate instantaneous bulk velocity distribution by area-averaging of individual sphere velocities. It is observed that the interstitial liquid can enhance bulk basal slip due to lubricating effect. The liquid viscosity also gives rise to a unifying effect on bulk dynamics: sharp interface between moving and stationary spheres can be detected in dry mass but not for the liquid-immersed mass. Lastly, we apply the Coulomb failure criterion developed in soil mechanics for homogeneous infinite static mass to predict the failure plane in current finite mass in unsteady motion. It is found that as long as the basal slip is taken into account, the failure plane can be successfully captured in the dry and the water-immersed avalanches.

9:44AM G11.00009 ABSTRACT WITHDRAWN —

9:57AM G11.00010 Characterizing titanium powder: morphology, flow and segregation[1] , NICHOLAS A. POHLMAN, JOHN A. ROBERTS, MATTHEW GONSER, Northern Illinois University — Powder metallurgy manufacturing with titanium could achieve near-net shape for parts with high strength-to-weight ratio. However, pure titanium (Ti) powders have large aspect ratios that prevent smooth flow. Features of Ti powder were examined experimentally to understand its limited flowability. The surface morphology was measured using a SEM for both raw powder as well as those made "uniform" via milling. The poly-disperse mixture had particle sizes between 45–700 μm, of which a significant concentration were $< 45\mu$m, which results in a very steep and unsteady angle of repose when the powder was placed in a rotating tumbler. The flat surface and steady flow typical of macroscopic particles is not present, but instead has slumping motion as material piles up and collapses intermittently. Spectral analysis indicates that increasing Froude number causes the slumping behavior to move to higher frequencies, but spread over a larger range. Size segregation causes porous streaks to form at the periphery of the tumbler where the angular separation of the streaks decreases with increasing Froude number. The overall conclusion is that more narrow ranges of particle size are necessary to yield usable titanium in powder metallurgy manufacturing.

[1]Funding provided by U.S. Army contract numbers W56HZV-04-C-0783, W56HZV-09-C-0656.

Monday, November 21, 2011 8:00AM - 9:57AM – Session G12 Turbulence Simulation IV 315

8:00AM G12.00001 Toward a simplified wall boundary condition for wall-modeled large eddy simulation[1] , JUNGIL LEE, MINJEONG CHO, HAECHEON CHOI, Seoul National University — In this study, we provide the mean wall shear stress as a boundary condition for wall-modeled large eddy simulation (WMLES) without any further modeling near the wall. The motivation of using this wall boundary condition for WMLES is that in the framework of finite volume method, the accurate information of mean wall shear stress may be most important in the momentum transport near the wall, even if the first grid used in WMLES locates far away from the wall ($y^+ = O(100) \sim O(1000)$), where y is the wall-normal distance. For turbulent channel flow, the mean wall shear stress is balanced with the mean pressure gradient and thus is *a priori* prescribed during WMLES. The results of WMLES at $Re_\tau = 2000$ and 20000 with this boundary condition show very good prediction. In general turbulent boundary layer flows, the mean wall shear stress is *a priori* unknown and thus a method of dynamically obtaining the mean wall shear stress during the computation will be suggested and examined.

[1]Supported by the WCU and BK21 Program, NRF, MEST, Korea.

8:13AM G12.00002 LES of channel flow with wall roughness , N. SAITO, D.I. PULLIN, California Institute of Technology, H.M. BLACKBURN, Monash University — We describe LES, using a spectral-element method, of turbulent channel flow with sub-grid wall roughness at moderate to large Reynolds numbers. The wall-model of Chung and Pullin (*JFM, 2009*) is extended to account for local subgrid roughness by the addition of a single empirical roughness function $u_\tau\,\Delta^+(k_s^+)$, where $k_s^+ = k_s\,u_\tau/\nu$ and k_s is the equivalent sand roughness. This term is used in both the inner-scaling ansatz used to reduce the unsteady term of the wall-normal integration of the stream-wise momentum equation and also in the derivation of the "log-like" profile used to provide the outer slip velocity, as a boundary condition for the LES, at a raised virtual wall. In these LES, standard expressions for $\Delta^+(k_s^+)$ of the Colebrook type are used. The LES produce results that include the mean velocity profile and some turbulence statistics as a function of Reynolds number for fixed ratio of k_s to the channel half height.

8:26AM G12.00003 Large Eddies in the Flow over Two-Dinemsional Dunes , MOHAMMAD OMIDYEGANEH, UGO PIOMELLI, Queen's University — We performed large-eddy simulation of the flow over two- dimensional large-scale river-bed irregularities called dunes at laboratory scale (the Reynolds number based on the average channel height and mean velocity is 18,900). The flow separates at the dune crest, generating a shear layer that plays a crucial role in the transport of momentum and energy, as well as the generation of coherent structures, and reattaches $5.2h$ downstream on the stoss side of the dune. Near-wall turbulence is strongly affected by the concave shape of the bed after the reattachment point cause streamwise Taylor-Gortler vortices to be generated. These vortices cause more coherent organized elongated streaks close to the wall in contrast to the streaks in turbulent boundary layer. The flow over dunes has unique characteristics among open channel flows because of the presence of spanwise vortices in the separated shear layer generated by the Kelvin- Helmholtz instability. Large eddies are formed when the spanwise vortices in the shear layer undergo lateral instabilities result into inclined horseshoe-like vortices. Between the legs of these vortices, strong ejections occur (with $u'v'$ over 40 times and turbulent kinetic energy over 15 times larger than the local mean). When these eddies reach the surface, the ejection causes a local stagnation point on the surface, with divergent streamlines, similar to those observed in the field when upwellings occur.

8:39AM G12.00004 Direct and Large Eddy Simulation of stably stratified turbulent Ekman layers , STIMIT SHAH, ELIE BOU-ZEID, Princeton University — Understanding and parameterizing turbulent fluxes in statically stable boundary layers, where buoyant forces destroy turbulent kinetic energy remains a challenging, yet very important problem, in geophysical fluid dynamics. Numerical simulations, with their ability to provide 3D, time-varying information on turbulent structures and dynamics are increasingly used to tackle the problem. However, some uncertainties remain about the limitation to low Reynolds number (Re) in Direct Numerical Simulation (DNS) and performance of subgrid-scale models in Large Eddy Simulation (LES) under stable conditions. To better understand these limitations and the effect of stability, we combine DNS of stably stratified Ekman layer at moderate Re ($\sim 10^{3-4}$ based on U_∞ and δ_t) with wall-modeled LES using Lagrangian scale-dependent dynamic (LASD) subgrid model at large Re ($\sim 10^{7-8}$). The analysis focuses on effect of stability on turbulent kinetic energy budget, turbulent structure, mean profiles of velocity, potential temperature, Reynolds stress, rms velocities and streamwise velocity and potential temperature spectra.

8:52AM G12.00005 A modulated gradient model for scalar transport in large-eddy simulation of the atmospheric boundary layer , HAO LU, St. Anthony Falls Laboratory, University of Minnesota, FERNANDO PORTE-AGEL, School of Architecture, Civil and Environmental Engineering, École Polytechnique Fédérale de Lausanne (EPFL) — As a simple alternative to the standard eddy-diffusivity closure, a nonlinear subgrid-scale (SGS) flux model is introduced and implemented in simulations of a neutral atmospheric boundary layer with a constant surface flux. This approach is based on the Taylor expansion of the SGS flux, and employs the local equilibrium hypothesis to evaluate the SGS velocity scale and the SGS scalar concentration scale. To resolve the instability issue of the original gradient model and ensure numerical stability, we adopt a clipping procedure to avoid local negative SGS production of the scalar variance. The model formulation using constant coefficients is assessed through a systematic comparison with well-established theoretical predictions and reference results of various flow statistics. Simulation results obtained with the use of this new model show good agreement with the reference results and a significant improvement compared to results obtained using traditional eddy-diffusivity models. For instance, the new model can deliver the expected surface-layer similarity (logarithmic) scalar profile and power-law scaling of the power spectrum of scalar fluctuation. Simulations using the new model also yield reasonable flow structures and scalar statistics.

9:05AM G12.00006 Large-eddy simulation of stationary homogeneous stratified sheared turbulence , GEORGIOS MATHEOU, DANIEL CHUNG, Jet Propulsion Laboratory/California Institute of Technology — Using a buoyancy-adjusted extension of the stretched-vortex subgrid-scale (SGS) model suitable for large-eddy simulation (LES) of stratified flows we investigate stationary and homogeneous shear-driven turbulence. The SGS model is free of parameters and is consistent with features of anisotropic mixing frequently observed in stratified flows. The vortex-based construction naturally constrains the mixing in the horizontal provided the vortex alignment is favorable even at high gradient Richardson numbers. By comparison with a corresponding set of high Reynolds number direct numerical simulations (DNS), the performance of the SGS model is assessed, with focus on its performance as the grid resolution approaches the overturning scale of the flow. Moreover, several aspects of stratified turbulence dynamics are discussed.

9:18AM G12.00007 Development of Filtered Density Function for Large Eddy Simulation of Entropy Transport in Turbulent Reacting Flows , MEHDI SAFARI, M. REZA H. SHEIKHI, Northeastern University — A new methodology based on filtered density function (FDF) is being developed for large eddy simulation of entropy transport in turbulent reacting flows. The filtered form of entropy transport equation includes several unclosed entropy generation terms. The closure is provided by the FDF, which represents the joint entropy, frequency, velocity and scalar probability density function within the subgrid. An exact transport equation is developed for the FDF. The unclosed terms in this equation are modeled by considering a system of stochastic differential equations incorporating the second law of thermodynamics. The modeled FDF transport equation is solved by a Lagrangian Monte Carlo method. The methodology is employed to simulate turbulent shear flows, involving transport of entropy. The predicted results are assessed by comparing with data generated by direct numerical simulation (DNS). The predictions show good agreements with the DNS data.

9:31AM G12.00008 Large Eddy Simulation of Shock / Isotropic Turbulence Interaction , NATHAN GRUBE, PINO MARTIN, University of Maryland — We compute large eddy simulations (LES) of shock / isotropic turbulence interactions (SITI) and compare to direct numerical simulation (DNS) results. The inflow turbulence is highly compressible with turbulence Mach numbers of 0.5 to 0.9, and the convective Mach numbers range from 3.5 to 5. The inflow turbulence is strong enough to significantly corrugate the main shock but still lies within the unbroken shock front regime. The Taylor microscale Reynolds numbers of the inflows are approximately constant in the range 22-26. Preliminary LES results show that the turbulence anisotropy created by the interaction is captured very well by the LES. The return of the turbulence to local isotropy (as measured by anisotropy of the vorticity fluctuations) is not yet well captured in the LES.

9:44AM G12.00009 SGS Modeling of the Internal Energy Equation in LES of Supersonic Channel Flow , SRIRAM RAGHUNATH, GILES BRERETON, Michigan State University — DNS of fully-developed turbulent supersonic channel flows ($Re_\tau = 190$) at up to Mach 3 indicate that the turbulent heat fluxes depend only weakly on Mach number, while the viscous dissipation and pressure dilatation do so strongly. Moreover, pressure dilatation makes a significant contribution to the internal energy budget at Mach 3 and higher. The balance between these terms is critical to determining the temperature (and so molecular viscosity) from the internal energy equation and so, in LES of these flows, it is essential to use accurate SGS models for the viscous dissipation and the pressure dilatation. In this talk, we present LES results for supersonic channel flow, using SGS models for these terms that are based on the resolved-scale dilatation, an inverse timescale, and SGS momentum fluxes, which intrinsically represent this Mach number effect.

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G13 Multiphase Flow III: Experiments 316

8:00AM G13.00001 The Breakup Mechanism and the Spray Pulsation Behavior of a Three-Stream Atomizer , CHIN HEI NG, University of Washington, ANNE DORD, GE Global Research, ALBERTO ALISEDA — In many processes of industrial importance, such as gasification, the liquid to gas mass ratio injected at the atomizer exceeds the limit of conventional two-fluid coaxial atomizers. To maximize the shear rate between the atomization gas and the liquid while maintaining a large contact area, a secondary gas stream is added at the centerline of the spray, interior to the liquid flow, which is annular in this configuration. This cylindrical gas jet has low momentum and does not contribute to the breakup process, which is still dominated by the high shear between the concentric annular liquid flow and the high momentum gas stream. The presence of two independently controlled gas streams leads to the appearance of a hydrodynamic instability that manifests itself in pulsating liquid flow rates and droplet sizes. We study the dependency of the atomization process on the relative flow rates of the three streams. We measure the size distribution, droplet number density and total liquid volumetric flow rate as a function of time, for realistic Weber and Ohnesorge numbers. Analysis of the temporal evolution of these physical variables reveals the dominant frequency of the instability and its effect on the breakup and dispersion of droplets in the spray. We present flow visualization and Phase Doppler Particle Analyzer results that provide insight into the behavior of this complex coaxial shear flow.

8:13AM G13.00002 Flash atomization in splash plate nozzles , REZA KARAMI, NASSER ASHGRIZ, Department of Mechanical and Industrial Engineering, University of Toronto, HONGHI TRAN, Chemical Engineering Department, University of Toronto — An experimental study was conducted to determine the effect of mass flow rate and temperature on characteristics of liquid sheets formed by splash plate nozzles. Two different splash plate nozzle sizes were used at five flow rates and 6 temperatures. The splash plate angel of the nozzles was 55 degrees and the working liquid was water. Direct visualization via photography was performed to investigate the sheet breakup process. Reynolds number and Jacob number were used to characterize the sheet formation and breakup regime. Four different sheet instabilities and breakup mechanisms can occur depending on the temperature and mass flow rate of the nozzle. The results were compared to the results obtained from other experiments using different liquid and nozzle size. The characterization can help to predict the sheet breakup mechanism at any condition.

8:26AM G13.00003 Classification of secondary atomization mechanisms , NICOLAS RIMBERT, Universite de Nancy — In the 20$^{\text{th}}$ century both drop towers and shock tubes have been used to study the breakup of an accelerated drop in an air strream. This led Faeth and colleagues to synthesize the different breakup modes observed in a chart involving the Weber *We* and the Ohnesorge *Oh* number of the droplet (*We* is the ratio between kinetic energy and surface energy of the droplet whereas *Oh* governs the damping rate of its surface oscillations). The most important drop breakup mechanisms are for increasing values of *We*: the bag breakup, the shear breakup and the so-called catastrophic breakup. Rayleigh-Taylor instability (RTI) is one of the main explanations and it has recently elucidated some experimental results with success, first thanks to accurate measures of the drop acceleration and then by estimating it through use of drag coefficients. The point is that the droplet firstly deforms into a spheroid which increases its drag coefficient and eventually accelerates the growth of RTI. Unlike most linearized theory, The Droplet Deformation and Breakup (DDB) model makes use of a variational technique to obtain the non linear evolution equation of the semi-axis of the spheroid. It will be shown how to combine both the DDB theory and either the classical RTI or a viscous extension. We will firstly show how the different domain can be theoretically derived. Then the classical hypothesis that the deformation mechanism is much quicker than the growth of the RTI will be proved invalid for high *We*.

8:39AM G13.00004 PTV implementation on two-phase flow in a forced impinging jet[1] , RAHUL MULINTI, KENNETH KIGER, University of Maryland - College Park — Two-phase flow experiments have been conducted to predict particle suspension and sedimentation within coupled particle-laden flows relevant to rotorcraft brownout conditions. A hybrid PIV/PTV technique has been implemented to improve the performance in high concentration regions, while still retaining the flexibility inherent to PTV to resolve multi-valued velocity displacements within a given interrogation region. These processing tools have been optimized and their reliability has been validated using synthetic particle images in a prescribed Taylor-Green vortex flow model. The parametric space of investigation included particle image density, Stokes number and image delay times. Experiments have been conducted to study the interaction of a mobile sediment bed with characteristic flow structures similar to those within a rotor wake. The mobilization conditions and wall-normal flux of particulates by the vortex-wall interaction will be reported for different particle size classes, and are correlated to the local vortex conditions such as vortex decay and its subsequent three dimensionalization. The effect of turbulent coupling between the particle and fluid momentum, as based on a point-particle drag law valid for dilute concentrations of particles has been examined.

[1]Work supported by AFOSR under grant FA9550-08-1-0406.

8:52AM G13.00005 Cavitation and bubble cloud dynamics in a high-intensity focused ultrasonic field , YUAN LU, JOSEPH KATZ, ANDREA PROSPERETTI , Department of Mechanical Engineering, Johns Hopkins University — We focus a high power (200 W max, 500 kHz) ultrasonic beam to generate cavitation in quiescent water, and observe the process using high-speed holography. The intense pressure fluctuations cause corresponding variations in water density and refractive index, generating diffraction patterns that are evident in the holograms. These enable us to map the instantaneous spatial structures of the pressure field, and resolve its mean features, such as pressure nodes in a partial standing wave. At low powers, we observe slow growth of bubbles and their migration to the nodes due to the primary Bjerknes force. At high powers, this process persists in the periphery of the focal zone, but bubble clouds grow explosively near the center of the beam. These clouds travel in the sound propagation direction at a velocity of up to 5 m/s, but slow down briefly at the pressure nodes, while growing and shrinking. The bubbles contain mostly vapor, as the clouds vanishes in <100 μ s when the sound is turned off. Likely causes for these phenomena include acoustic streaming, Bjerknes forces, and attenuation of sound in the clouds.

9:05AM G13.00006 Wind tunnel experiments on the interactions between turbulence and small inertial droplets[1] , COLIN BATESON, ALBERTO ALISEDA, University of Washington — Understanding the dynamics of particles in turbulent flows is important to many engineering and environmental problems including spray atomization as well as cloud- droplet growth and precipitation. Specifically, we have studied the effect of turbulence on droplet collision-coalescence in an effort to clarify its role in the process of warm rain formation. We are exploring the hypothesis that turbulence-induced-collisions can explain the size gap between the limit of condensational growth and the onset of gravitational collisions and sedimentation. We use wind tunnel experiments to study the evolution of water droplets in homogeneous, isotropic, slowly decaying grid turbulence. We analyze the process of preferential concentration and the enhanced relative velocity of droplets in the 1-100 micron range due to their inertial interactions with the underlying turbulence. We collect droplet size and velocity data from a Phase Doppler Particle Analyzer (PDPA) to understand the influence of turbulence on the droplet collision kernel, and to quantitatively model it in terms of the Radial Distribution Function. We use high-speed visualizations to obtain two-dimensional droplet velocity fields to validate the PDPA point measurements, and to model the relative velocity distribution as a function of droplet pair spacing and Stokes number.

[1]Supported by NSF grant ATM-0731248

9:18AM G13.00007 Turbulent boundary layer on a large freely moving particle suspended in high-Reynolds number isotropic turbulence , MARGARET BYRON, COLIN MEYER, GABRIELE BELLANI, EVAN VARIANO, University of California, Berkeley — We experimentally measure the flow surrounding neutrally buoyant particles freely moving in homogeneous, isotropic turbulence. Particle size is within the inertial subrange of the ambient turbulence. We measure the flow using stereoscopic particle image velocimetry which resolves 3 velocity components in a nearly 2-dimensional planar volume. We calculate ensemble average velocity statistics, conditional on the particle kinematics. Two-point statistics indicate the effect of the moving particle on the surrounding turbulent flow. We use these results to consider the impact of ambient turbulence on aquatic microorganisms, specifically those which are larger than the Kolmogorov timescale and small enough that swimming does not dominate transport.

9:31AM G13.00008 Time Resolved 2D X-Ray Densitometry of a Ventilated Partial Cavity Closure[1] , SIMO MAKIHARJU, STEVEN CECCIO, University of Michigan — A time resolved x-ray densitometry system was developed to measure the spatial distribution of void fraction for nominally two-dimensional flows. The system can image a region of (15 cm)^2 at a frame rate of up to 4000 fps. The source was a rotating anode type normally used for cineradiography and angiography. Supplied by a 65 kW high frequency generator with a high speed starter, it could be operated at up to 433 mA at 150 kV. The imager subsystem comprised of a high speed camera coupled with a high resolution image intensifier. The range of measured void fraction can be changed to span a desired range yielding an uncertainty on the order of 1% of the measurement range. The system is used to examine the void fraction field in the closure region of a ventilated partial cavity behind a backward facing step. The cavity has Reynolds number of $O(10^5)$ based on the cavity length, and a non-dimensional gas flux of $Q^* = 0.0048$. The bubbly flow created in the cavity wake is examined using the x-ray densitometry system, duel fiber optical probes, and high speed cinematography. The local void fraction and bubble size distributions in the cavity wake are determined, and the measurements methods are compared.

[1]The research was sponsored by ONR under grant N00014-08-1-0215, program manager Dr. L. Patrick Purtell.

9:44AM G13.00009 Dynamics of single rising bubbles in a liquid-solid system , NASIM HOOSHYAR, ROBBERT F. MUDDE, PETER J. HAMERSMA, Delft University of Technology, SANKARAN SUNDARESAN, Princeton University, J. RUUD VAN OMMEN, Delft University of Technology — While the dynamics of single rising gas bubbles in clear liquids has been studied extensively, the dynamics of bubbles in liquid-solid slurries containing small particles ($d_s < 100\mu$m) is not yet well understood. We have investigated the rise characteristics of single gas bubbles ($< d_b >$=3mm) in a neutrally buoyant suspension of Polystyrene particles ($< d_s > = 78\pm5.79\mu$m) in a glycerol-water mixture. Using a four-point optical probe we have obtained information on the bubble motion and shape in the slurry. The velocity of single bubbles, rising at terminal velocity through the suspension, slightly decreases with increasing solids volume fraction; this was accompanied by a change in the shape of the bubbles from oblate ellipsoidal to nearly spherical. The viscosity and the surface tension of the liquid in the presence of suspended particles were also measured. Our results show that the viscosity was almost doubled with a 20% increase in the solids volume fraction. We found that the rise velocity of a bubble in the slurry is close to that in a pseudo-single phase liquid with the same viscosity and surface tension.

9:57AM G13.00010 Experimental Validation of Dam-Break Problem for Two-Phase Flow Numerical Simulation

, KEISUKE HARA, ZENSAKU KAWARA, TOMOAKI KUNUGI, Department of Nuclear Engineering, Kyoto University, TAKU NAGATAKE, Japan Atomic Energy Agency — In order to validate the numerical method it is important to compare the results of the numerical simulation to the experimental ones. The dam-break problem is one of the well-known validation tests for the multiphase flow computations. However, a few experimental data are available until now regarding the problem. In the validation test, it is necessary to consider the differences between the experiment and the numerical simulation. In this study, we conducted the dam-break experiments and measured the position of the water-front and the gravity center of the water region. These experimental results were compared with the numerical results obtained by the MARS (Kunugi, 2001). The water-front position has been used as a reference to validate the numerical method. When the size of the liquid column changed to double without changing the aspect ratio, the non-dimensional water-front positions were changed. On the other hand, the gravity center positions of the water region obtained by the experiments for both cases were in good agreement with each other and also with the numerical results. As a conclusion, the gravity center position must be used as the validation measure instead of the water-front position. Kunugi,T. CFD Journal, 9, (2001) pp.563-571

Monday, November 21, 2011 8:00AM - 10:10AM –

Session G14 Flow Control II 317

8:00AM G14.00001 Low Reynolds Number Flow Dynamics of a Thin, Flat Airfoil with Elastically Mounted Leading Edge Actuator[1]

, SOURABH APTE, Oregon State University — Direct numerical simulations are performed to investigate the effect of an elastically mounted leading edge actuator on the unsteady flow at high angles of attack over a flat, thin airfoil at Reynolds number of 14700 based on the chord length. The leading edge actuator is mounted with a torsion spring at one-third the chord length allowing dynamic variations in the effective angle of attack through flow-induced oscillations. The goal is to investigate potential benefits of flow induced flapping motion of the leading edge actuator to the lift and drag characteristics of thin airfoils. The structural model for the rigid actuator is based on a torsional spring-mounted compound pendulum. A fictitious-domain based finite volume approach [(Apte et al. (JCP 2009)] is used to compute this fluid-structure interaction problem on a fixed background mesh. It is shown that a lock-in region leading to limit cycle oscillations of the leading edge actuator can be achieved for certain spring parameters leading to improvements in mean lift-to-drag ratio.

[1]This work was supported under ASEE SFFP program at Wright Patterson Airforce base under the guidance of Dr. Miguel Visbal.

8:13AM G14.00002 Passive control of the flow in a symmetric channel with a sudden expansion based on linear stability analysis

, ANDREA FANI, SIMONE CAMARRI, MARIA-VITTORIA SALVETTI, University of Pisa — In the present work we investigate the stability properties of the flow in a 2D-plane channel with a symmetric sudden expansion, which is a possible schematization of a plane diffuser. The laminar flow in 2D diffusers may produce either symmetric or nonsymmetric steady solutions, depending on the value of the Reynolds number Re as compared with some critical value Re_c. The stability properties of the flow are studied in the context of linear theory, characterizing the sensitivity of the instability to both a structural perturbation of the linearized flow equations and a perturbation of the base flow. The information provided by the two kinds of analysis is used to propose a passive flow control strategy, obtained by the introduction of a small cylinder in the channel, aimed at stabilizing the symmetric solution at $Re > Re_c$. The effectiveness of this control strategy in the nonlinear case is investigated, and ultimately verified by numerical simulations of the flow in which a real control cylinder is introduced. The robustness of the control is also tested by numerical simulations in which the position of the cylinder is slightly changed with respect to the optimal one given by the sensitivity analysis.

8:26AM G14.00003 Simultaneous achievement of drag reduction and heat transfer augmentation in wall turbulence by optimal control theory

, AKIRA YAMAMOTO, YOSUKE HASEGAWA, NOBUHIDE KASAGI — The analogy concept between momentum and heat transfer, which is based upon the similarity between the Navier-Stokes and energy transport equations, has been widely used for analyzing turbulent transport phenomena. This fact implies inherent difficulty in enhancing heat transfer with pumping power reduced or not increased as much as heat transfer. We consider a fully developed turbulent channel flow with uniform heat generation in the fluid, so that the averaged transport equations have an identical form. The problem thus posed offers the most difficult challenge to achieve dissimilar momentum and heat transfer. Even under such a difficult condition, we demonstrate that the dissimilar control is possible if we exploit the continuity constraint on the velocity field. In order to optimize the control input, i.e., wall blowing/suction, the optimal control theory developed by Bewley et al. (2001) is applied. By defining the cost functional as a weighted sum of the intensity of the wall blowing/suction and the analogy factor, i.e., the ratio of wall heat flux and skin friction, the analogy factor increases more than double beyond that in the uncontrolled flow. It is also found that the resultant optimized control input exhibits a downstream traveling wave-like property.

8:39AM G14.00004 Robustness of reduced-order observer-based controllers in transitional 2D Blasius boundary layers

, BRANDT BELSON, Princeton University, ONOFRIO SEMERARO, KTH Royal Institute of Technology, CLARENCE ROWLEY, Princeton University, JAN PRALITS, University of Salerno, DAN HENNINGSON, KTH Royal Institute of Technology — In this work, we seek to delay transition in the Blasius boundary layer. We trip the flow with an upstream disturbance and dampen the growth of the resulting structures downstream. The observer-based controllers use a single sensor and a single localized body force near the wall. To formulate the controllers, we first find a reduced-order model of the system via the Eigensystem Realization Algorithm (ERA), then find the H_2 optimal controller for this reduced-order system. We find the resulting controllers are effective only when the sensor is upstream of the actuator (in a feedforward configuration), but as is expected, are sensitive to model uncertainty. When the sensor is downstream of the actuator (in a feedback configuration), the reduced-order observer-based controllers are not robust and ineffective on the full system. In order to investigate the robustness properties of the system, an iterative technique called the adjoint of the direct adjoint (ADA) is employed to find a full-dimensional H_2 optimal controller. This avoids the reduced-order modelling step and serves as a reference point. ADA is promising for investigating the lack of robustness previously mentioned.

8:52AM G14.00005 Turbulent drag reduction by transverse wall oscillations

, RASHAD MOARREF, MIHAILO R. JOVANOVIC, University of Minnesota — Skin-friction drag reduction by transverse wall oscillations has received significant attention in the last two decades. Both experiments and simulations have demonstrated that oscillations with properly selected amplitude and frequency can reduce turbulent drag by as much as 40 percent. For a turbulent channel flow, we develop a model-based approach to design oscillations that suppress turbulence. We show that judiciously selected linearization of the flow with control can be used to determine turbulent eddy viscosity in a computationally efficient way. The resulting correction to the turbulent mean velocity is then used to identify optimal frequency of oscillations, which is in close agreement with previously conducted experimental and numerical studies. This demonstrates the predictive power of our simulation-free approach to controlling turbulent flows.

9:05AM G14.00006 A computational framework for adjoint-based study of flapping wings[1]

, MINGJUN WEI, New Mexico State University — The study of flapping wings has provided many challenges for numerical simulation and experimental measurement by the problem's unsteadiness, nonlinearity, moving boundary, fluid-structure interaction, and other factors. However, even with perfect flow field from perfect numerical simulation, there is still limited understanding of fundamental flying mechanism. The sensitivity information, as a higher order quality, shows sometimes the mechanism often hidden in direct numerical simulation and experiments. Adjoint-based method, by its nature, suits the best for solving sensitivity problems with large input space and small output space. The work presented here is to provide a general framework to apply adjoint-based method to solve the sensitivity of flapping-wing problems. Immersed boundary technique is used in dealing with the moving boundaries in both physical and adjoint spaces.

[1]Thanks to the support by AFRL Summer Program and AHPCRC (ARL)

9:18AM G14.00007 Control Performance of Input Shaping to Reduce Liquid Sloshing in a Horizontally Accelerating Container

, DONGJOO KIM, SEONG-WOOK HONG, KYOUNGJIN KIM, Kumoh National Institute of Technology — Input shaping was originally developed to suppress the structural vibrations and many successful results have been reported so far. Its applicability can be extended to fluid vibrations such as liquid sloshing, which occurs when a partially filled container experiences acceleration for fast positioning control. However, its performance for sloshing suppression has not been assessed by solving the governing equations for unsteady two-phase fluid motions. Therefore, the objective of this study is to numerically investigate the performance of input shaping to reduce liquid sloshing in a horizontally accelerating container. In this study, three different input shapers (ZV, ZVD, convolved ZV shapers) are considered to eliminate the sloshing completely. With all of the shapers investigated, liquid sloshing is successfully suppressed but the control effects are quite dependent on the input shaper. In the cases of ZV(Zero Vibration) and ZVD(Zero Vibration and Derivative) shapers, liquid sloshing is suppressed to some extent, but residual vibrations still remain. On the other hand, liquid sloshing is almost eliminated with a two-mode convolved ZV shaper, which is designed to suppress the most energetic (first and third) modes.

9:31AM G14.00008 Flow past a moderately heated horizontal cylinder at low Reynolds number

, A. SAMEEN, S. AJITH KUMAR, Aerospace engg, IITMadras, Chennai, India, S. ANIL LAL, Department of Mechanical Engineering, ollege of Engineering Trivandrum, India — In the present work, the influence of heat on vortex shedding and the wake structure in a flow cylinder is analyzed. Flow past a heated two dimensional cylinder in the laminar regime is investigated in the range of Reynolds numbers 25 to 100 under varying buoyancy conditions, $0 \leq Ri \leq 1.0$ (where Ri is the Richardson number, indicates the relative dominance of the inertial and buoyant effects). A hybrid FEM-FVM method is used to solve Navier-Stokes and energy equations. The vortex shedding frequency defined non-dimensionally as Strouhal number is observed to be increasing with heating. Vortex shedding behind heated cylinder exhibit asymmetry in wake region due to stable and unstable density stratification below and above the cylinder respectively. A negative lift is generated due to heating, in an otherwise symmetric zero time-averaged lift configuration, due to skewness in vortex shedding towards the direction of buoyancy. We also note that the heating brings down the drag slightly for the cases considered here. We also note here that at large Ri, the drag coefficient (C_D) decrease by $\sim 30\%$ at $Re = 25$, percentage of decrease being less pronounced at higher Re (=100).

9:44AM G14.00009 On the evaluation of control performance in drag reducing flows[1]

, YOSUKE HASEGAWA, BETTINA FROHNAPFEL, Center of Smart Interfaces, TU Darmstadt, MAURIZIO QUADRIO, Department of Aerospace Engineering, Polytechnic Institute of Milan — Performance of drag reduction control for internal flows is customarily evaluated under two flow conditions, i.e., constant mass flow rate or constant pressure gradient. In the former, the drag reduction rate, i.e. the reduction of the pumping power, is used for evaluating control performance, while the increase rate of the mass flow rate indicates successful control in the latter. Considering real applications, however, the optimization problem of flow control is essentially formulated by the interplay between energy saving and achieved flow rate, that can be interpreted as money on one side, and time or convenience on the other side. Based on this idea, we derive two dimensionless parameters which quantify the costs of total energy consumption and convenience for transportation of fluid through a given duct. We reevaluate the control performances of existing strategies in the present framework and also show that this evaluation can easily be extended to external flows.

[1]The authors acknowledge support of FR2823/2- 1 and 3-1 of the German Research Foundation (DFG), and the Japanese Society for the Promotion of Science (JSPS) Postdoctoral Fellowship for Research Abroad.

9:57AM G14.00010 A reduced order interaction model for flow control via a poroelastic carpet of compliant actuators

, DIVYA VENKATARAMAN, ALESSANDRO BOTTARO, University of Genoa, Italy, RAMA GOVINDARAJAN, Jawaharlal Nehru Centre for Advanced Scientific Research, Bangalore, India — A passive actuation technique that consists of covering the suction side of an airfoil with a coating of porous, compliant actuators (analogous to a class of flight feathers known as "coverts"), has in the recent past been shown to be effective in controlling its wake. A feature in the case of an airfoil (as opposed to the case of a cylinder) is that two flow frequencies are observed under many conditions. Such wake control is highly sensitive to whether or not the frequency of the structure's dynamics synchronizes with one of these frequencies. Possible transitions from non-chaotic to chaotic vortex shedding regimes have also been observed. To determine the physics governing this complex fluid-structure interaction problem, a reduced order model is developed, by coupling an oscillator model for the structure with a van der Pol-like nonlinear oscillator model for the vortex shedding of the airfoil (a form with which the vortex-shedding from many canonical structures has been represented). In the case of a cylinder, the limit cycle of the unsteady lift coefficient for the coupled system is seen to lie within that of the plain vortex-shedding oscillator, indicating that the poroelastic coating is capable of yielding drag reduction, arising from a stabilization of the wake.

Monday, November 21, 2011 8:00AM - 10:10AM – Session G15 Free-Surface Flows I 318

8:00AM G15.00001 A regularised one-dimensional drop formation and coalescence model on a single Eulerian grid

, THEO DRIESSEN, University of Twente, ROGER JEURISSEN, Eindhoven University of Technology, HERMAN WIJSHOFF, OCE Technologies NV, ARJAN VAN DER BOS, JACCO SNOEIJER, DETLEF LOHSE, University of Twente — Droplets with a well-controlled size and speed are required in many industrial and medical applications. In this work we study axisymmetric droplet formation from a piezo inkjet print head. The breakup of an axisymmetric viscous jet is considered in the lubrication approximation. The discretised equations are solved on a fixed equidistant one-dimensional Eulerian grid. The governing equations are implemented in a conservative second order accurate total variation diminishing (TVD) scheme, preventing the numerical diffusivity. Singularities that occur at pinchoff and coalescence are regularised by a small modification on the surface tension. The modification is of the order of the spatial step. This regularisation ensures that the solution of the presented numerical model converges to the exact solution of the breakup of a jet in the lubrication approximation. The results of the presented numerical model agree quantitatively with the analytical solution of the Rayleigh-Plateau instability, and with experimental results on the final stage of the Rayleigh-Plateau instability.

8:13AM G15.00002 Splash and droplet ejection after disc impact on a liquid , IVO R. PETERS, DEVARAJ VAN DER MEER, University of Twente, The Netherlands, JOSE MANUEL GORDILLO, Universidad de Sevilla, Spain — When a circular disc hits a water surface, a thin sheet of liquid is ejected close to the edge of the disc. Surface tension deforms the tip of this sheet into a rim which, depending on the experimental conditions, becomes unstable and ejects small droplets. At the region very close to the edge of the disc and for times close to the moment of impact, we find that the liquid sheet shows a clear self-similar behavior for any value of the Weber number provided that $Fr = U^2/(gR_{disc}) > 1$. We demonstrate, by using both experimental results and boundary-integral simulations, that the ejection of droplets in this experiment is caused by a Rayleigh-Taylor instability. This results from a strong downwards pointing acceleration present in the liquid sheet, much larger than the gravitational acceleration. By determining the length-scale and the acceleration at the tip of the sheet, we show that the transition to droplet ejection occurs when the Bond number is of order 1.

8:26AM G15.00003 Surface Seal in the Water-Entry of Hydrophobic Spheres , BENJAMIN JACKSON, SUNGHWAN JUNG, PAVLOS VLACHOS, Virginia Tech — We examine the surface seal phenomenon observed in the water entry of hydrophobic spheres. Using high speed shadowgraph imaging we experimentally investigate the dependence of surface seal time and critical pressure on projectile size, density, and impact velocity. In the initial stage of impact the projectile slams into the free surface projecting a splash curtain upward. As the projectile descends into the fluid, an air cavity forms behind it and eventually pinches off from the atmosphere. Previous studies have primarily focused on the low velocity case where surface seal does not occur and the events above the free surface are often ignored. As the projectile velocity increases, the pressure drop within the cavity increases. Surface seal occurs when this pressure drop exceeds a critical pressure and is characterized by the closure of the splash curtain. The splash curtain domes over and seals the cavity above the free surface. This dome closure emits a downward jet which propagates into the air cavity from above and affects the later cavity dynamics. We present scaling arguments for critical pressure and surface seal time based on our observations.

8:39AM G15.00004 The Nearfield of a Shallow Angle Plunging Jet - Periodicity of Air Cavity Formation , SURAJ S. DESHPANDE, MARIO F. TRUJILLO, University of Wisconsin - Madison — The plunging of a water jet into a quiescent water pool is investigated computationally, using the Volume-of-Fluid methodology in the framework of the open source utilities of OpenFOAM. For the shallow angle plunges ($\theta <\sim 20^\circ$), our computations and previous experiments at Dynaflow Inc. revealed a distinct periodicity in the formation of large air cavities in the nearfield. In this work, we analyze the periodicity and present a closed form expression to describe it. Our analysis, based on potential flow treatment, regards the cavity as a Rankine body [Oguz et al., JFM 1995] and its motion as that of a standing wave. For the jets considered ($Fr \sim O(10)$, $Re \sim O(10^5)$, $We \sim O(10^3)$ and $\theta <\sim 20^\circ$.), the frequency of cavity formation was found to be related to the jet diameter and gravity, independent of viscous and surface tension effects. Our analysis, which is valid only for shallow plunges, is in excellent agreement with this observation. For steeper jets, we demonstrate that this periodicity vanishes and small air cavities are chaotically drawn into the pool.

8:52AM G15.00005 Eggs in Milk: The Conclusion[1] , KEN LANGLEY, JEFF HENDRICKS, MATTHEW ELVERUD, DAN MAYNES, TADD TRUSCOTT, Brigham Young University — A hard-boiled egg spinning on a countertop and passing through a puddle of milk draws milk up the side of the egg and then ejects it at the maximum radius. This same phenomenon occurs for any partially submerged spinning object whose radius increases upward from the fluid surface (e.g., spheres, inverted cones, rings, etc.). In particular, spheres are used to investigate the behavior of this phenomenon and its sensitivity to experimental parameters. Three modes of ejection – jets, sheets, and sheet break-up – are identified, which are highly dependent on several parameters: viscosity, angular velocity, immersion depth of sphere, and sphere diameter. Experimental results are presented with comparisons to a theoretical model that is derived using integral conservation of momentum. This phenomenon can be used as a pump to easily remove fluids from shallow areas.

[1]BYU ORCA

9:05AM G15.00006 Angled wing air induction for microbubble drag reduction[1] , YUICHI MURAI, Hokkaido University, ICHIRO KUMAGAI, YUJI TASAKA, YOSHIAKI TAKAHASHI, RAND Engineering Co. Ltd — Interfacial dynamics above an angled wing which is submerged into shallow water is investigated. Our experimental study aims at designing a high-performance bubble generator for microbubble drag reduction in marine vehicles. The performance being parameterized by the size and the amount of bubbles is determined by flow physics which is represented by triple interference among the air layer, the water flow, and the solid wing. The wing gives rapid deformation of the interface as well as disturbance before downstream high-speed wave-breaking and further later bubble fragmentation with help of Kelvin-Helmholtz instability. We have already demonstrated several practical installation of the device onto commercial ships from small to large scale. The presentation deals with the visualization of the wing-above behavior of gas-liquid interface that triggers the generation of fine bubbles in its downstream layer.

[1]NEDO and JSPS

9:18AM G15.00007 DNS prediction of the properties of solitary waves running down a vertically falling film[1] , P.-K. NGUYEN, V. BONTOZOGLOU, Dept. of Mechanical Engineering, University of Thessaly, 38334 Volos, Greece, S. CHAKRABORTY, C. RUYER-QUIL, Laboratoire FAST, UMR CNRS 7608, Campus universitaire, 91405 Orsay, France — The present work computes the properties of stationary, traveling wave by solving the Navier-Stokes equation using the Galerkin finite element method. Solitary-like waves are derived by considering a long enough computational domain and necessarily a strong mesh refinement in the vicinity of the solitary wave in order that the properties of the wave converge asymptotically to the true solitary limit. The solitary wave can be characterized by its maximum height, and phase speed in terms of the modified Reynolds number, $\delta = 3^{4/3}Re^{11/9}Ka^{-1/3}$. Both the wave height and the phase speed curves exhibit inflection points in the transition region, then maxima at intermediate values of δ, and finally a drop to a plateau at high enough δ. These are unique characteristics of the full second-order model by Ruyer-Quil and Manneville. In particular, the behavior deep in high δ limit shows that (i) the phase speed tends asymptotically to a value almost unaffected by Ka, and (ii) the wave height increases roughly linearly with Ka, which underlines the stabilizing effect of viscous diffusion at low Kapitsa numbers.

[1]Research partially supported by the EU/FP7-ITN-MULTIFLOW

9:31AM G15.00008 Propagation of axisymmetric capillary waves along a periodically-pinned liquid cylinder: Bloch waves and scattering[1] , LIKUN ZHANG, DAVID THIESSEN, Washington State University — An open capillary channel consisting of a water-filled helical wire is useful for phase separation because of its ability to capture impacting droplets. Capillary wave packets are generated by the impact and transport energy from the impact zone. In order to understand the propagation characteristics, we develop a theoretical model for a simplified axisymmetric channel consisting of a liquid cylinder with the free surface pinned by a periodic array of rings. The theory is based on the Bloch theorem and an eigenfunction expansion. From the semi-analytical solution we have computed the band structure, dispersion relation, phase speed, group speed, and energy flux, as well as their dependence on channel parameters. Results are interpreted in terms of the single-ring scattering properties and compared to multiple scattering for finite ring arrays.

[1]Supported by NASA

9:44AM G15.00009 Scattering of axisymmetric capillary waves on a liquid cylinder by a pair of surface-pinning rings and the relation to liquid-bridge resonance[1] , DAVID THIESSEN, LIKUN ZHANG, Washington State University — We consider the scattering of time-harmonic axisymmetric capillary waves on an infinite inviscid liquid cylinder by two infinitesimal coaxial surface-pinning rings. A semi-analytical solution is obtained by solving the truncated system of equations based on an evanescent wave expansion. The computations find that there is an infinite set of discrete frequencies at which the transmission is total, and these frequencies are shown to correspond to the resonant frequencies of the liquid bridge that spans the rings. These frequencies are close to the resonant modes of an inviscid liquid bridge between two impenetrable disks. An approximate solution is also obtained for the case of widely spaced rings using single-ring scattering properties. The extension of the approximate method to the computation of capillary mode resonances for other geometries will be discussed.

[1]Supported by NASA

9:57AM G15.00010 Field-enhanced electrokinetic charge separation induced by thermocapillarity-driven flow in a liquid film , MATHIAS DIETZEL, STEFFEN HARDT, Institute of Nano- & Microfluidics, Center of Smart Interfaces, TU Darmstadt — Thermocapillary stresses acting tangentially to the interface of a thin liquid film in a wide slot subjected to a lateral thermal gradient induce a rotational flow pattern. We demonstrate analytically and numerically for an electrolyte as working fluid that this flow can be used to separate charges and induce a streaming potential based on the electrokinetic effect, converting thermal to electric energy. The charge separation efficiency is commonly spoiled by the circumstance that the excess ions accumulate in direct vicinity of the wall where frictional losses are highest. As an alternative, we impose a traverse electric field across the film so that excess ions also accumulate at the gas-liquid interface where the highest flow velocities occur. No power is needed to maintain the external field if the deformation of the interface remains negligibly small, requiring in turn sufficiently small applied thermal and electric gradients below critical values of hydrodynamic and electrostatic instabilities. We estimate the increase and the upper limit of the streaming potential as well as of the overall conversion efficiency as a function of the applied voltage and the film thickness scaled to the Debye length.

Monday, November 21, 2011 8:00AM - 9:57AM – Session G16 Air Vehicle Aerodynamics 319

8:00AM G16.00001 High Speed Focused Schlieren Flow Visualization of Shock Induced Dynamic Stall[1] , PATRICK BOWLES, DUSTIN COLEMAN, THOMAS CORKE, FLINT THOMAS, University of Notre Dame, MARK WASIKOWSKI, Bell Helicopter — Focused Schlieren images of the leading edge flowfield ($0 \leq x/c \leq 0.10$) of a modern helicopter rotor-blade airfoil are presented in order to document the development of shock waves during compressible light dynamic stall at moderate subsonic freestream Mach numbers ($M_\infty \in$ [0.2,0.6]). The focused Schlieren system was designed to provide a "macro" image of the near (suction) surface boundary layer and shock features with a marginal depth of focus, effectively eliminating density distortions associated with the windtunnel (plexiglass) walls or the (polycarbonate) rotating airfoil endplates. A high-speed camera operating at up to 4 kFPS was used to capture the dynamic behavior of the shock, leading-edge separated shear layer and dynamic-stall vortex during the pitching cycle. These images were then correlated with static pressure time series on the airfoil surface. The effect of leading-edge roughness was then investigated. Under certain conditions, this was shown to alter the shock development and shock streamwise propagation. The effect that the shock formation had on the pitch-moment stability for these different conditions is then presented.

[1]Supported by Bell Helicopter

8:13AM G16.00002 Boundary Layer Trip Effects On Incipient Dynamic Stall Flutter[1] , THOMAS CORKE, PATRICK BOWLES, DUSTIN COLEMAN, FLINT THOMAS, University of Notre Dame, MARK WASIKOWSKI, Bell Helicopter — Experimental results of compressible light dynamic stall on a modern helicopter rotor-blade airfoil are presented at free-stream Mach numbers ranging from 0.2 to 0.6, Reynolds numbers up to 3.5×10^6, and reduced frequencies corresponding to relevant helicopter rotor speeds ($k \leq 0.05$ for $M_\infty \geq 0.4$). Diagnostic tools include high frequency pressure transducers to study the temporal behavior of the pressure field and integrated loads, as well as macro focused Schlieren images of the leading edge flow field ($0 \leq x/c \leq 0.10$). The study concentrates on the air load's net aerodynamic work on the airfoil to identify combinations of collective and cyclic input that cause negative or reduced torsional damping. Free (un-tripped) and forced (tripped) laminar to turbulent transition were explored in order to constrain the mechanism for dynamic stall onset. Leading edge roughness resulted in a suppression of the suction peak and strong dynamic vortex compared to the baseline that resulted in reduced torsional damping at high subsonic free-stream Mach numbers ($M_\infty \geq 0.5$). Comparisons to predictions from a Leishman-Beddoes model for dynamic stall are also presented.

[1]Supported by Bell Helicopter

8:26AM G16.00003 Velocity measurements in the near wake of a ship superstructure[1] , CODY BROWNELL, LUKSA LUZNIK, MURRAY SNYDER, HYUNG SUK KANG, U.S. Naval Academy, COLIN WILKINSON, Zenetex LLC — Velocity measurements in a ship airwake are obtained *in situ* aboard a 108 ft naval training vessel. Three-component sonic anemometers are placed at the bow of the ship, for reference wind measurement, and at numerous locations above a flight deck at the stern of the ship. The mean flow structure resembles that of a 3D backward-facing step, with a recirculation region covering much of the forward flight deck, and significant downwash over the remainder of the surface. Reynolds stresses and two-point velocity correlations are presented, and placed in the context of shipborne helicopter operations. The influence of an atmospheric boundary layer, often unavailable in ship airwake measurements from a wind tunnel, is discussed.

[1]Support from the ONR Young Investigator Program (PI: MR Snyder) is gratefully acknowledged.

8:39AM G16.00004 Energy Exchange during Plunge/Surge Motions of a 2D Wing[1] , WESLEY KERSTENS, Illinois Institute of Technology, JEESOON CHOI, TIM COLONIUS, California Institute of Technology, DAVID WILLIAMS, Illinois Institute of Technology — The rate of energy transfer between an NACA-0006 wing and an unsteady flow is examined at pre-stall and post-stall conditions using numerical simulations and wind tunnel experiments. The plunge and surge motions simulate the fluctuating vertical (w_z) and longitudinal (w_x) velocity components of a wind gust. In a steady flow the wing loses energy to the flow through the drag power term, but in an unsteady flow the wing may gain energy from the fluctuating lift power and fluctuating drag power terms. The net energy transfer averaged over the period of oscillation depends on the phase angle between the plunge and surge motions. The largest increase of energy occurs when w_x and w_z are in-phase. When the fluctuations are large enough, then it is possible for the net energy gain to be positive. The numerical simulations conducted at Reynolds numbers near the critical value for vortex shedding show qualitative agreement with the experiments. The simulations highlight the role of vortex shedding in determining the optimal frequency and phase for energy extraction from the gust.

[1]Support of the AFOSR through grant FA9550-09-1-0189 managed by Dr. Douglas Smith is gratefully acknnowledged.

8:52AM G16.00005 Facility Dependent Issues Influencing the Experimental Characterization of a Laminar Separation Bubble[1] , DAVID A. OLSON, AHMED M. NAGUIB, MANOOCHEHR M. KOOCHESFAHANI, Michigan State University — There is a documented discrepancy in the characteristics of the laminar separation bubble on the steady SD7003 airfoil obtained from various experimental and computational studies. The influence of added freestream turbulence, among other factors, is studied for a range of Reynolds number (2×10^4 – 6×10^4), and angle of attack (2° – 12°). Both the baseline cases and those with added grid-generated turbulence utilize single-component Molecular Tagging Velocimetry with a cross-stream measurement spatial resolution of 52μm. The addition of freestream turbulence is shown to decrease the size of the separation bubble by both delaying separation and triggering an earlier reattachment. The measured separation location is shown to be also sensitive to the uncertainty of wall location and near-wall spatial resolution.

[1]This work was supported by AFRL through the Michigan/AFRL Collaborative center in Aeronautical Sciences and, in part, by AFOSR grant number FA9550-10-1-0342.

9:05AM G16.00006 Experimental Investigation of Tip Vortex Control Using a Half-Delta Shaped Tip Strake , JENNIFER PEREIRA, McGill University — Due to their undesirable lift-induced drag effects, the control of tip vortices remains a challenge for the aviation industry. Naturally, most types of control devices involve tip vortex alleviation and thus focus on altering the flow at the wingtip. One such device is a half-delta shaped strake that when added to the wingtip could not only reduce induced drag by improving end effects but also increase lift through the delta wing LEV lift. To investigate and better understand this concept a rectangular planform NACA0012 wing was fitted with a 65° sweep half-delta shaped tip strake and force measurements were obtained at a chord Reynolds number of 2.7×10^5. This data was complimented with seven-hole probe flowfield measurements over the tip and in the near field. Results were compared to the rectangular planform wing (baseline wing), a sharp 65° sweep half-delta and full delta wing for direct comparison and to better understand the flow physics involved. The effect of both streamwise location, angle of attack and strake setting were examined. It was found that the strake acts much like a delta wing albeit with a strengthened LEV that prematurely breaks down. The broken down LEV then results in a tip vortex which is much more diffused than its baseline counterpart. In order to quantify this effect, the Maskell model was used to calculate the induced drag of both the baseline wing and the strake at various angles of attack and strake settings.

9:18AM G16.00007 Drag reduction in large wind turbines through riblets: evaluation of different geometries , ROGER ARNDT, LEONARDO CHAMORRO, FOTIS SOTIROPOULOS, University of Minnesota — Achieving skin friction drag reduction by use of riblets has been a topic of intensive research throughout the last several decades. The majority of the effort on this topic has been based on both numerical (mainly DNS) and experimental (wind tunnel and fluid channel) approaches. Yet, despite these valuable endeavors, the fundamental mechanisms that induce the drag reduction are still not well established. In this study, wind tunnel experiments were performed to quantify the drag reduction for a wind turbine airfoil caused by different V-grooved riblet configurations. A full-scale 2.5MW Clipper wind turbine airfoil section (of 1 meter chord length, typical of the 88% blade span) was placed in the freestream flow of the wind tunnel at the Saint Anthony Falls Laboratory, University of Minnesota. The drag forces the airfoil experienced were measured for different riblet configurations and at different angles of attack, all with a constant Reynolds number of Re=2.2 millions (based on the airfoil chord length). Layouts of both complete and partial airfoil coverage, of riblets, were considered in the study. Force sensors were used to measure Lift and Drag but more accurate Drag forces were obtained through wake surveys using a pitot static probe. The measurements will be used to help develop and test the performance of near-wall boundary conditions in the context of RANS and hybrid RANS/LES models.

9:31AM G16.00008 Aerodynamic Behavior at One Revolution Angle of Attack of Two-Dimensional Wings[1] , YONG OUN HAN, School of Mechanical Engineering, Yeungnam University, EUN HA LEE, Graduate Student, JEONG HYUN KIM, Research Associate, YONG OH SHIN, WESTEC — In order to investigate aerodynamic behaviors at extreme angles of attack beyond the normal static stall angle and in the reversed flow, lift and drag have been measured at one revolution angles of attack by rotating the wing around the 1/4 chord with use of a dynamic balance in the low speed wind tunnel. Three different geometries of wing section; a flat plate, a symmetric airfoil, NACA0018, and a cambered airfoil, Goe222, were selected for these experiments. It was turned out that the lift coefficient maintains substantially even beyond the traditional stall AoA of the wing. Drag coefficients of these wings showed sinusoidal profiles, and polar plots of Cl versus Cd provided distinctive behaviors unseen in the calculation by the classical wing theory. Application of the cyclic aerodynamic characteristics to a vertical axis wind turbine and wake characteristics around the critical angle will be displayed.

[1]This work was supported by Cooperative R&D program between Industry, Academy, and Research Institute funded Korea Small and Medium Business Administration in 2011.

9:44AM G16.00009 Electro-Luminescence based Pressure-Sensitive Paint System for Unsteady Flow Field Measurements , YOSHIMI IIJIMA, Aerospace Research and Development Directorate, JAXA, HIROTAKA SAKAUE — Electro-luminescence (EL) based pressure-sensitive paint (PSP) system is developed for capturing unsteady flow fields. It has advantages in uniform distribution in the illumination without remotely apply the illumination source from the testing object. The resultant system can be applied directly onto a testing object surface. It consists of an inorganic EL and a PSP. The EL emits blue illumination uniformly applied onto the PSP layer. Because of a sheet illumination, the EL gives uniform distribution, while a point illumination gives a spot in illumination. The PSP is developed to provide a fast response to a change in pressure. It uses a porous particle and a polymer to create porous-polymer PSP. The response time characterization of the developed system is included in the presentation. The developed system is applied to an unsteady flow field, such as a sound field in a resonance tube.

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G17 Waves II 320

8:00AM G17.00001 The Non-Linear Response of a Water Free Surface to a Pressure Distribution Moving at Constant Speed , N. MASNADI, J.H. DUNCAN, University of Maryland, T.R. AKYLAS, Massachusetts Institute of Technology — The non-linear response of a water free surface to a localized pressure distribution moving at constant speed just below the minimum phase speed (C_{min}) of gravity-capillary waves is studied experimentally. The experiments are performed in a tank that is 6 m long and 0.25 m wide with water depths ranging from 15 to 40 mm. The pressure distribution is generated by blowing air on the water surface via a vertically oriented 2-mm-ID tube that is mounted on an instrument carriage. The bottom of the tank is made of clear plastic and covered with translucent paper. A checkerboard pattern is printed on the paper and this pattern is viewed from above the water surface with a high-speed digital movie camera. The images of the checkerboard pattern are distorted by refraction at the water free surface and yield qualitative observations and quantitative measurements of the temporal evolution of the wave pattern. At towing speeds close to but below C_{min}, a gravity-capillary lump appears behind the pressure source. For higher speeds, but still below C_{min} an unsteady wave pattern consisting of a wide V is generated. At the boundary between the two response states, the pattern is asymmetric in the cross-stream direction. Above this boundary, lumps are shed from the tips of the V and the frequency of shedding increases with towing speed.

8:13AM G17.00002 Numerical simulation of wind-wave evolution and breaking , YI LIU, LIAN SHEN, ROBERT DALRYMPLE, Johns Hopkins University — The wind-wave evolution and breaking play an important role in air-sea interaction by affecting the mass, momentum, and energy exchange between the atmosphere and the oceans. We perform large-eddy simulation of wind-wave interaction using a coupled level set/volume-of-fluid (CLSVOF) method. The dynamic evolution of waves including the entire breaking process under the wind forcing is simulated. With CLSVOF method, surface overturning, pinching off, and merging associated with wave breaking are captured. Different wind speeds are considered and their effects on the wave growth and breaking are examined. Based on the simulation results, various wind-wave interaction stages including wave growth, evolution, and breaking are investigated. Spectral analysis and wavelet analysis are applied to the wave field. The wave spectrum evolution under wind forcing is quantified and frequency downshift is observed. Wave groups leading to breaking are identified. The influence of the airflow separation over breaking waves on wave growth is also examined.

8:26AM G17.00003 Laboratory study of the wind structure over surface waves , MARC BUCKLEY, FABRICE VERON, University of Delaware — Airflow dynamics above waves strongly influence exchanges of heat, momentum and mass between the Ocean and the Atmosphere. We present experimental results on the detailed structure of the airflow above waves. The experiments took place at University of Delaware's large (42m long, 1m wide, and 1.25m high) wind-wave facility where a variety of winds, wave ages and steepnesses were generated by a wind-tunnel and a mechanical wave generator. Airflow properties within and above the viscous sublayer were obtained using PIV, while wave profiles and spectra were measured by laser-induced fluorescence. We intermittently observe a separation of the viscous sublayer past the wave crest in certain wind-wave conditions. Despite the intermittent aspect of this sheltering effect, when averaged over all wave phases, our results suggest that there is a substantial along-wave variability of the surface viscous tangential stress, which in turn may affect wave growth and the air-water momentum balance.

8:39AM G17.00004 Nonlinear growth of wind-driven oblique surface waves by critical-layer interaction , SANG SOO LEE, Naval Surface Warfare Center, Carderock Division — Nonlinear interaction between two (primary and secondary) free-surface waves of the same streamwise phase velocity and wind is studied by extending the linear resonant theory of Miles (1957). A nonlinear interaction occurs when the normalized amplitude of the primary 3-D wave becomes of the order of the cube of the density ratio of air to water. If the secondary wave (of smaller amplitude) is also an oblique wave, the nonlinear coupling between them generates a difference mode. Its amplitude becomes as large as the secondary wave in air, but smaller than the latter in water. In addition, the nonlinear interaction between the primary and difference modes induces a parametric growth on the secondary wave if the primary frequency is higher than the secondary frequency. The primary wave remains linear. The secondary and difference mode amplitudes are governed by integro-differential/integral equations. Numerical results show that the nonlinear growth rates become much larger than the linear growth rates. This nonlinear coupling occurs when the waves are very small. The nonlinear interaction is entirely confined to a thin critical layer (in air), and the perturbations outside the critical layer is governed by linear dynamics. It is shown that the initial nonlinear growth of a free-surface wave is governed by the mode-mode interaction in air rather than in water.

8:52AM G17.00005 Airflow over groups of water waves , SHAR SAJJADI, Embry-Riddle Aeronautical University, JULIAN HUNT, University College London — An analytic in conjunction with a numerical model is developed for the turbulent shear flow of an airflow over steadily moving wave groups in which individual waves are unsteady. Firstly this shows, by linear theory, for the individual waves in the group the combined effects of the unsteady critical layer flow, and the viscous/turbulent sheltering on the lee sides of the wave. Secondly, by using weakly non-linear theory to analyze the disturbed air flow over the waves in groups, it is shown how the air speed over the downwind part of the group is lower over the upwind part. This asymmetry causes the critical layer height to be lower over the downwind part, which leads to critical-layer-effect producing a net horizontal force on the waves, in addition to the sheltering effect. This analysis, which is confirmed by numerical solutions, shows why the critical layer is present over monochromatic waves but does not produce a net force, despite earlier arguments to the contrary. The concepts based on this analysis indicate how the effects of wave dynamics also affect the wave growth.

9:05AM G17.00006 Kinetic Energy Balance Perspective on the Structure and Dynamics of the Air Flow Over Surface Waves , TIHOMIR HRISTOV, Dept. of Mechanical Engineering, Johns Hopkins University — The transition from current empirical to mechanistic understanding of the interaction between wind and the wavy water surface requires describing the spatio-temporal dynamics of the wave-induced fields in the wind. Studies have reported failure to detect dynamic or statistical signature of such fields in spectra, structure functions, kinetic energy balance, or in turbulent dissipation measurements. Here we analyze the wave-induced fields of velocity and pressure, their structure functions and higher order moments, e.g. energy and momentum fluxes. The existence of wave-induced fields is demonstrated. While velocity is predominantly turbulent, pressure is organized and coherent with the waves. The latter indicates that random force mechanism for wind-wave interaction proposed by C. Eckart (1953) and O.M. Phillips (1957) is inefficient and virtually inactive. The explicit form of the structure function obtained here and the partitioning of fluxes into turbulent and wave-induced, explain the failures to observationally detect a wave signature in the structure function and in the TKE budget.

9:18AM G17.00007 Numerical study of interaction of turbulence with free surface and wave , XIN GUO, LIAN SHEN, Johns Hopkins University — Direct numerical simulation is performed for free-surface turbulent flows. The Navier-Stokes equations subject to fully nonlinear dynamic and kinematic free-surface boundary conditions are integrated in time using a fractional-step method. The kinematic boundary condition is advanced in time with a Runge-Kutta scheme to obtain the evolution of surface elevation. Pseudo-spectral and finite-difference methods are used respectively in the horizontal and vertical directions for spatial discretization. Computational grid is clustered towards the free surface to ensure that flow details near the free surface are captured adequately. Various Froude and Weber numbers are considered. Surface signatures, e.g. propagating waves and surface roughness, are illustrated. The effect of free surface and surface waves is found to be important to the underlying turbulence. The variations of turbulence statistics and characteristic vortical structures near the free surface are elucidated. The budgets of turbulence kinetic energy and enstrophy are also quantified and their dependence on the free-surface conditions is analyzed.

9:31AM G17.00008 Spatial statistics of capillary wave turbulence , MICHAEL BERHANU, ERIC FALCON, MSC, Paris Diderot University — Wave turbulence concerns the study of the statistical properties of a set of numerous nonlinear interacting waves. The archetype of this phenomenon are waves on the surface of a fluid [1]. One of the challenge of wave turbulence is to reach a simultaneous 2D-space and time measurement of the wave amplitude that goes further than usual spatially localized ones. Recently, Herbert et al. [2] performs this measurement for gravity wave turbulence using an optical method (Fourier Transform Profilometry). Here, we used another optical technique (Diffusing Light Photography [3]) with a fast camera, to reach a better spatial resolution that allows us to investigate spatiotemporal statistics of capillary wave turbulence. Linear and nonlinear dispersion relations of capillary waves are obtained as well as the full spectrum of wave amplitude both in the k and omega spaces. Statistical properties are extracted and analyzed in order to test the validity of Weak Turbulence Theory. Moreover, as this technique is not limited to small amplitudes of surface deformation, we can observe appearance of small capillary waves on steep gravity waves, and to check the impact of this gravity to capillary conversion on the wave turbulence.

[1] A. C. Newell and B. Rumpf, Annu. Rev. Fluid Mech. 43, 59 (2011) [2] E. Herbert, N. Mordant and E. Falcon, Phys. Rev. Lett. 105, 144502 (2010) [3] W. B. Wright, R. Budakian and S. J. Putterman, Phys. Rev. Lett. 76, 4528 (1996)

9:44AM G17.00009 Rain-induced momentum exchange at the ocean surface under low wind speed conditions , EMILY HARRISON, FABRICE VERON, University of Delaware — We present results from laboratory experiments on the generation of turbulence and the damping of wind-waves by rainfall. These experiments were conducted in a wind-wave flume that is 7.3 m long, 0.48 m wide, and 0.61 m tall, with the mean water level maintained at 0.4 m. A rain module, 0.86 m by 0.38 m, was suspended 4.98 m above the flume to allow droplets to reach approximately 91% of terminal velocity at impact. Turbulence was measured using both particle image velocimetry and planar laser induced fluorescence. Rain effects on the wave field were investigated with an optical wave gauge directly beneath the rain. We have completed a series of 70 experiments encompassing 7 rain kinetic energy flux conditions, 5 wind speeds, and 2 salinities (0 and 37ppt). We find that rainfall generates intense turbulence and mixing. We show that the mixing occurs in bursts associated with vortex rings generated by single drops, which are then sheared by the background current. This shearing and loss of coherence in the vortices generated by the drop impacts prevents significant vortex pairing thereby limiting the depth of the mixing region to a few 10s of cm. The depth of this mixed region is further reduced when raining on salt water.

9:57AM G17.00010 Experiments on Normal Modes in a Tank with Corrugated Bottom , ANDRZEJ HERCZYNSKI, Boston College, PATRICK WEIDMAN, University of Colorado, Boulder, LOUIS HOWARD, MIT, JIE YU, North Carolina State University — We report an experimental investigation of standing waves in a rectangular container of corrugated bottom partially filled with water. The study was stimulated by the theory of Howard and Yu[1,2] predicting the existence of resonant normal modes — called Bragg resonance — wherein the amplitude grows exponentially, either from one end of the channel to the other or from the center out in each direction, depending on the endwall phases of the corrugated bottom. The tank, with adjustable length between 450 and 490 cm, width 13 cm, and height 30 cm, is fitted with a sinusoidal bottom (made of high density polyurethane foam) of wavelength 52 cm and peak-to-peak amplitude 5 cm. Waves are excited by shaking the container in periodic horizontal motion using an electrical motor. The amplitude of the standing waves was recorded using two sensitive pressure probes and also observed and filmed through the transparent acrylic walls. Experimental results are in essential agreement with the theory.

[1] *J. Fluid. Mech.* (2007), *vol.* 593, *pp.* 209-234.
[2] *J. Fluid. Mech.* (2010), *vol.* 659, *pp.* 484-504.

Monday, November 21, 2011 8:00AM - 9:57AM –
Session G18 Microfluids: General IV 321

8:00AM G18.00001 Thermally-Induced Mixing of Microscale Droplets for Detection of Gas-Phase Analytes , MEYSAM BARMI, CHRYSAFIS ANDREOU, MEHRAN HOONEJANI, BRIAN PIOREK, MARTIN MOSKOVITS, CARL MEINHART, University of California Santa Barbara — In this talk, we investigate the absorption and mixing kinetics inside of sessile droplets, subject to a specified temperature gradient and evaporation rate. Such microscale droplets have free-surface interface that enables the adsorption of airborne molecules and their subsequent identification by Surface Enhanced Raman Spectroscopy (SERS). A droplet of SERS-active colloidal suspension can be mixed efficiently with the adsorbed analyte due to the Marangoni effect. This stimulates an aggregation process that results in the creation of the so called "SERS hotspots." The droplets are interrogated using a Raman spectrometer to obtain highly sensitive and specific analyte detection. Efficient mixing is critical for practical measurements in many chemical and biological applications. We investigate specified thermal gradients and evaporation rates for precision mixing, and optimize these parameters for high-performance vapor detection. Numerical simulations using COMSOL Multiphysics are used to investigate droplet dynamics, by predicting fluid motion and analyte-induce aggregation kinetics. For the velocity field inside of the sessile droplet, Micro-PIV experiments are used to validate the numerical simulations. The system is tested using gas phase 4-Aminobenzenethiol as a model analyte.

8:13AM G18.00002 Monodisperse droplet generation for microscale mass transfer studies[1] , CHRISTINE ROBERTS, REKHA RAO, ANNE GRILLET, CARLOS JOVE-COLON, CARLTON BROOKS, MARTIN NEMER, Sandia National Laboratory — Understanding interfacial mass transport on a droplet scale is essential for modeling liquid-liquid extraction processes. A thin flow-focusing microfluidic channel is evaluated for generating monodisperse liquid droplets for microscale mass transport studies. Surface treatment of the microfluidic device allows creation of both oil in water and water in oil emulsions, facilitating a large parameter study of viscosity and flow rate ratios. The unusually thin channel height promotes a flow regime where no droplets form. Through confocal microscopy, this regime is shown to be highly influenced by the contact angle of the liquids with the channel. Drop sizes are found to scale with a modified capillary number. Liquid streamlines within the droplets are inferred by high speed imagery of microparticles dispersed in the droplet phase. Finally, species mass transfer to the droplet fluid is quantitatively measured using high speed imaging.

[1] Sandia National Laboratories is a multi-program laboratory operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85

8:26AM G18.00003 Stationary microdroplets in a Hele-Shaw Cell: anchoring mechanism and flow field , SUNGYON LEE, Mathematics, UCLA, RÉMI DANGLA, LadHyX and department of Mechanics, Ecole Polytechnique, FRANÇOIS GALLAIRE, Mechanical Engineering, EPFL, CHARLES BAROUD, LadHyX and department of Mechanics, Ecole Polytechnique — When a droplet is confined inside a Hele-Shaw cell, it adopts a "pancake"-like shape from its relaxed spherical state, resulting in increase in its surface energy. Variations in the level of confinement lead to the gradient in surface energy, or a force, to "anchor" droplets against the mean external flow. In this talk, we discuss the experimental and theoretical findings of this novel anchoring mechanism. In addition, a peculiar three-dimensional flow pattern has been observed on the boundary of the anchored drop. We show the robustness of this flow pattern over various flow conditions and provide the theoretical justification based on the Marangoni stresses on the droplet interface.

8:39AM G18.00004 Flow patterns of microbubble streaming in microfluidic settings , CHENG WANG, SHREYAS V. JALIKOP, SASCHA HILGENFELDT, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Steady streaming flows from oscillating microbubbles have demonstrated a number of promising applications in microfluidics, including manipulating microparticles, mixing enhancement, transporting liquid and lysing vesicles. Earlier experimental studies have shown that oscillating bubbles located on or close to solid boundaries produce a variety of streaming flow patterns, such as "fountain" and "anti-fountain" flows. These flow patterns depend on the viscosity of the liquid, and on the driven frequency and amplitude. Here we take advantage of the micro-fabrication technology to create microbubbles of controlled size, position and also with different exposure to solid boundaries to study bubble streaming. Our new experimental results show that the flow topology depends not only on the frequency and amplitude, but also significantly on the solid boundaries near the bubbles. The findings are of importance towards the fundamental understanding of bubble streaming flows and helpful on guiding practical microfluidic device designs as well.

8:52AM G18.00005 Bubble shrinkage and growth: A solution to carbon dioxide dissolution and solubility , MILAD ABOLHASANI, EUGENIA KUMACHEVA, AXEL GUENTHER, University of Toronto — Dynamic and equilibrium aspects of carbon dioxide transport across gas-liquid interfaces impact a wide range of technical, physiological and geological applications. We investigate carbon dioxide transport by continuously guiding a train of uniformly sized carbon dioxide bubbles and non-saturated liquid segments through a microchannel. The bubbles initially shrink and later expand. We quantitatively link the evolution of the bubble size to kinetic and equilibrium characteristics of carbon dioxide dissolution. While the initial velocity of carbon dioxide bubbles and the length of liquid segments significantly affect the dissolution of carbon dioxide, these parameters cannot be externally imposed, due to the dynamic nature of microscale gas-liquid flows. We use an automated microfluidic platform (gas impermeable) in combination with an image-based feedback control strategy to keep the dependent parameters constant and systematically determine the rate of carbon dioxide dissolution and the equilibrium solubility of carbon dioxide-liquid mixtures for a wide range of pressures, temperatures and liquids in a flowable format.

9:05AM G18.00006 Dissolution without shrinking: the Epstein-Plesset problem in a channel flow , SUIN SHIM, JIANDI WAN, HOWARD STONE, Princeton University — The dynamics of dissolution of CO2 bubbles in microfluidic channels is studied experimentally and theoretically. The results show first a rapid dissolution regime followed by a second apparent equilibrium regime where the bubble radius is constant. We observed that, regardless of the surfactant concentration, bubbles stopped shrinking after ∼30ms following generation. In the equilibrium regime, the bubble sizes are larger at low concentration of surfactants than at high concentrations. We interpret the results by considering the pressure variation along the microfluidic channel and modify the Epstein-Plesset model for bubble dissolution. Our modified model with a time dependent pressure term as the bubbles move along the channel explains the transient and steady behaviors of CO2 bubbles in a channel flow. In particular, the model rationalizes why the bubbles continue to dissolve even though the bubble radius is constant, since the pressure change accompanying bubble translation causes a shape change compensating the shrinkage due to dissolution.

9:18AM G18.00007 Cusps and spouts in microfluidic systems , AURÉLIEN DUBOIN, Laboratoire de Microfluidique, MEMS et Nanostructure UMR Gulliver CNRS-ESPCI 7083, 10 rue Vauquelin 75005 Paris, France, FLORENT MALLOGGI, Laboratoire Interdisciplinaire sur l'Organisation Nanométrique et Supramoléculaire, CEA Saclay, 91191 Gif sur Yvette Cedex, France, FABRICE MONTI, PATRICK TABELING, Laboratoire de Microfluidique, MEMS et Nanostructure UMR Gulliver CNRS-ESPCI 7083, 10 rue Vauquelin 75005 Paris, France — By injecting mineral oil (inner phase) and polymer solutions (outer phase), in a microfluidic flow focusing geometry, we observed the formation of cusps. These cusps undergo a transition from a steady state, to a thin cylindrical spout (oil in polymer). These oil spouts, do not touch the walls, and are surprisingly stable (they do not break into droplets). We study the nature of the cusp-spout transition, and find it is of first order. By taking advantage of the stability of the jet, we expect to synthesize micro-wires with this approach.

9:31AM G18.00008 Formation of Satellite and subsatellite droplets in a flow-focusing junction for viscoelastic fluids , DENIS FUNFSCHILLING, Laboratoire Réactions et Génie des Procédés (CNRS UPR 3349), ODILE CARRIER, HUAI-ZHI LI — The formation of a cascade of satellite and subsatellite oil droplets is observed in a flow-focusing microfluidic junction (250 micrometer of characteristic length) in the presence of surfactant (Sodium Dodecyl Sulfate), and polymer (100 to 1000 ppm of PAAm of high molecular mass). The size and distribution of the satellite and subsatellite droplets is quite reproducible. One and only one satellite droplet is formed in the dripping regime in case of Newtonian fluids [1]. When PAAm is added to the solution, the viscosity becomes viscoelastic and satellite droplets are many. The mechanism of breakup leading to multiple satellite droplets is self-repeating, as observed in previous work [2, 3]. At low frequency, the number of satellite droplet can go up to 7 or more. The distribution is generally very structured: a unique mother satellite droplet is surrounded by two daughter droplets, each of these daughter droplet is surrounded again by two grand-daughter droplets so that there are 4 daughter droplets in total. The ratio in volume between each generation is about 30. [1] Funfschilling D., Debas H., Li H.-Z.and Mason T. G., 2009, Phys. Rev. E, 80, 015301 [2] Muzzio F. J., Tjahjadi M. and Ottino J. M. 1991, Phys. Rev. Lett. 67, 54-57 [3] Tjahjadi M., Stone H. A. and Ottino J. M. 1992, J. Fluid Mech. 243, 297-317

9:44AM G18.00009 Capillary focusing in the microfluidic Hele-Shaw channel , ALEXANDER LESHANSKY, Dept of Chemical Engineering, Technion-IIT, YULI HALUPOVICH, Taub Computer Center, Technion-IIT, LEN PISMEN, Minerva Center for Nonlinear Physics of Complex Systems, Technion-IIT — For many droplet-based mircofluidic applications it is desirable to create monodisperse droplets with adjustable volume and production rate directly in the microfluidic device. In this work we address the recently developed "step emulsification" technique that relies on an abrupt change in the aspect ratio of a single shallow and wide microchannel that merges into a deep reservoir. Such geometry yields a rapid and well controlled destabilization of confined co-flowing streams of immiscible liquids at the step separating the channel and the reservoir forming highly monodisperse droplets of the size comparable to the channel depth. The distinctive feature of the process is the capillary focusing effect, whereas the interface between the two fluids takes the shape of a tongue narrowing in the flow direction just ahead of the step; at the step the tongue tip becomes unstable generating droplets. We present theoretical modeling of the capillary focusing effect. The developed small-capillary-number asymptotic theory and the results of VOF numerical simulations show a good agreement with the experiments and provide some important insights into the underlying physics of the phenomenon.

Monday, November 21, 2011 8:00AM - 10:10AM – Session G19 Rayleigh Taylor Instability I 322

8:00AM G19.00001 Miscible experiments on the Rayleigh-Taylor instability using planar laser induced fluorescence visualization , MATTHEW MOKLER, MICHAEL ROBERTS, JEFFREY JACOBS, The University of Arizona — Incompressible Rayleigh-Taylor instability experiments are presented in which two stratified miscible liquids having Atwood number of 0.2 are accelerated in a vertical linear induction motor driven drop tower. A test sled having only vertical freedom of motion contains the experiment tank and visualization equipment. The sled is positioned at the top of the tower within the linear motors and accelerated downward causing the initially stable interface to be unstable and allowing the Rayleigh-Taylor instability to develop. Experiments are presented with and without forced initial perturbations produced by vertically oscillating the test sled prior to the start of acceleration. The interface is visualized using a 445nm laser light source that illuminates a fluorescent dye mixed in one of the fluids. The resulting fluorescent images are recorded using a monochromatic high speed video camera. The laser beam is synchronously swept across the fluorescent fluid, at the frame rate of the camera, exposing a single plane of the interface allowing for the measurement of spike and bubble mixing layer growth rates.

8:13AM G19.00002 Experiments on Rayleigh-Taylor instability with Multi-modal initial conditions at low Atwood numbers , SARAT CHANDRA KUCHIBHATLA, DEVESH RANJAN, Texas A&M University — The water channel facility at TAMU is employed for performing Rayleigh-Taylor experiments at low Atwood numbers ($\sim 10^{-3}$) using hot and cold water as working fluids. Effects of initial conditions on the onset of instability can be investigated using the novel flapper mechanism as previously presented. This mechanism using a very accurate, repeatable servo motor can induce multi-modal initial conditions consisting of up to 8 component modes. Visualization is performed using Nigrosene dye as marker with a LED backlight setting. Experiments were performed to observe the development of instability and its dependence on initial condition (wavelength and phase). Images of the flow-field for different multimodal initial conditions are presented here, and mixing layer growth estimated using ensemble-averaging technique. A measure of the molecular mixing (B_0, B_2 and θ from the BHR model of turbulence) in these flows was also performed using high-speed thermocouple measurements in the flow domain. Furthermore, bubble and spike velocities for single-mode initial experiments were measured using a seeded flow system and high-speed digital camera imaging. Velocities and growth rates are compared with Goncharov's analytical model.

8:26AM G19.00003 Effect of shear on R-T mixing at low and medium Atwood numbers ,

BHANESH AKULA, DEVESH RANJAN, Department of Mechanical Engg., Texas A&M University — Combined RT and KH instabilities are studied at three different Atwood numbers using the gas channel facility at TAMU. In the experiment, two gas streams of different densities (heavy over light) flowing parallel to each other are initially separated by a thin splitter plate. At the end of the splitter plate the two fluids are allowed to mix and the Rayleigh-Taylor instability develops. Simultaneous 3 wire and cold wire anemometry (S3WCA) is used to measure velocities and densities at different locations along and across the channel. Temperature is used as a marker to identify the streams and density is calculated from the temperature measured using a temperature probe with a constant current anemometer. High resolution digital imaging is performed to measure mixing heights and growth rates by injecting smoke into one of the streams and collecting the scattered light from the fog particles illuminated by the back lighting of the channel. Experiments are performed at Atwood numbers 0.04, 0.1 and 0.3. At these Atwood numbers, effect of shear is studied by varying the velocity of one the top stream. Initial conditions are characterized at the interface right after the splitter plate using S3WCA. Different parameters obtained from these measurements including, molecular mixing parameter θ, u', v', w' rms profiles, velocity correlations, vertical turbulent mass flux ρ'v' and their effect on mixing growth rate are discussed.

8:39AM G19.00004 An Update on Experiments and Simulations of Rayleigh-Taylor Instability in Elastic-Plastic Solids ,

PAMELA ROACH, ARINDAM BANERJEE, Missouri S&T — When an elastic-plastic plate is accelerated by a fluid of lower density, Rayleigh-Taylor instability (RTI) instability is mitigated by the mechanical strength of the plate. Such instances of RTI is observed in explosive welding, volcanic island formation, and inertial confinement fusion. In contrast to Newtonian fluids, experimental study of RTI in accelerated solids is traditionally hindered by difficult to measure material properties and exceedingly small time scales. Both experiments and simulations are used to define the instability region - moreover, the dependence of RTI on initial conditions in elastic-plastic solids is explored. In the experiment, a horizontally mounted disc is rotated to accelerate the interface between an elastic-plastic solid and air. The instability threshold and perturbation growth rate are captured using backlit imaging and a high speed camera. Two dimensional simulations are performed using finite element model ABAQUS to predict instability for a given initial perturbation and acceleration. Experiment and numerical simulation results are compared to the relevant literature.

8:52AM G19.00005 Progress on the Two-Wheel High Acceleration Experiment to Study Rayleigh-Taylor Turbulence[1] ,

AARON HALEY, ARINDAM BANERJEE, Missouri S&T — A new two-wheel experiment, scaled by a factor of 4 from the previously presented proof of concept, is used to study turbulent incompressible Rayleigh-Taylor (RT) instability. Two counter rotating wheels are mounted side by side such that axes of rotation are normal to gravity. A test section containing pairs of either miscible or immiscible fluids is attached to the first wheel and rotated so that a stable stratification is formed. The test section is then transferred to the adjacent wheel using a pneumatically actuated transfer mechanism. RT instability is effected by the inverted density stratification relative to the centrifugal acceleration. Late time RT turbulence at buoyancy $Re \approx 230,000$ is achieved. Details of the mixing layer development and growth constants are captured using high speed backlit imaging. A variety of fluid combinations (immiscible and miscible) are utilized to investigate development of RT mixing over a range of Atwood numbers and results are compared with data available in the literature.

[1]Funded by NSF-CBET-Fluid Dynamics Grant # 0967672 and DOE Los Alamos National Laboratory subcontract # 123141.

9:05AM G19.00006 Miscible and immiscible liquid experiments and simulations on the Rayleigh-Taylor instability ,

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

9:18AM G19.00007 ABSTRACT WITHDRAWN —

9:31AM G19.00008 ABSTRACT WITHDRAWN —

9:44AM G19.00009 On Accounting for Initial Conditions in Models of Rayleigh-Taylor Turbulent Mixing ,

BERTRAND ROLLIN, MALCOLM ANDREWS, Los Alamos National Laboratory — For many fluid engineering applications, considering only the fully developed turbulence regime with a rough estimate for the turbulence model initial conditions is not sufficient. If we consider for example Inertial Confinement Fusion (ICF), the turbulence and turbulent mixing induced by the Rayleigh-Taylor (RT) instability are subject to initial condition (IC) effects during the time of interest. The degree of confidence in any prediction of the flow evolution in the turbulence regime is therefore open to question. To improve our predictive capability, we need to capture the evolution of the instability from the initial state until it is appropriate to use a turbulence model, in particular with the treatment of ICs that seed the instability. We present our approach for tracking the growth of the RT mixing layer evolution. We have constructed a modal model based on several existing descriptions for single mode and multimode RT mixing evolution. We also present how to extract profiles of turbulence variables that are used as initial conditions for a turbulence model. Finally, we discuss a metric for switching from our modal model to the turbulence model.

9:57AM G19.00010 Surfactants and the Rayleigh-Taylor instability of Couette type flows ,

A.L. FRENKEL, D. HALPERN, A.S. SCHWEIGER, The University of Alabama — We study the Rayleigh-Taylor instability of slow Couette- type flows in the presence of insoluble surfactants. It is known that with zero gravity, the surfactant makes the flow unstable to longwave disturbances in certain regions of the parameter space; while in other parametric regions, it reinforces the flow stability (Frenkel and Halpern 2002). Here, we show that in the latter parametric sectors, and when the (gravity) Bond number Bo is below a certain threshold value, the Rayleigh-Taylor instability is completely stabilized for a finite interval of Ma, the (surfactant) Marangoni number: $\mathrm{Ma}_L<\mathrm{Ma}_1<\mathrm{Ma}<\mathrm{Ma}_2$. For $\mathrm{Ma}<\mathrm{Ma}_L$, the instability is longwave: the finite interval of unstable wavenumbers borders on the zero value. For $\mathrm{Ma}>\mathrm{Ma}_2$, and also for $\mathrm{Ma}_L<\mathrm{Ma}<\mathrm{Ma}_1$, the instability is "midwave": the interval of unstable wavenumbers is bounded away from both the zero and infinity. By numerical and asymptotic means, we determine typical dispersion curves and also characteristic dependencies such as the critical Marangoni numbers Ma_L, Ma_1, and Ma_2 as functions of the Bond number. We note that (for an interval of the Bond number) there are two distinct criticalities with nonzero (and distinct) critical wavenumbers.

Monday, November 21, 2011 8:00AM - 10:10AM –

Session G20 Interfacial/Thin Film Instability III 323

8:00AM G20.00001 On the Resistance of Nanofibrous Superhydrophobic Coatings to Hydrostatic Pressures[1] , T.M. BUCHER, B. EMAMI, H. VAHEDI TAFRESHI, M. GAD-EL-HAK, G.C. TEPPER, Virginia Commonwealth University — We present a numerical study aimed at investigating the influence of microstructural parameters on the resistance of submerged fibrous superhydrophobic coatings to elevated hydrostatic pressures. In particular, we generate 3-D virtual geometries comprised of randomly or orthogonally oriented horizontal fibers with bimodal diameter distributions resembling the microstructure of coatings produced via DC and AC electrospinning, respectively. These virtual geometries are then used as the computational domain for performing Full Morphology (FM) simulations to establish a relationship between the coatings' critical pressure—pressure beyond which the surface departs from the Cassie state—and their microstructures. Our numerical simulations are aimed at providing guidelines for the design and optimization of the coatings' microstructures.

[1]Financial support from DARPA, contract number W91CRB-10-1-0003, is acknowledged.

8:13AM G20.00002 Comparison of Navier-Stokes solution and lubrication approximation: dewetting of nanoscale liquid rings , SHAHRIAR AFKHAMI, LOU KONDIC, NJIT, JAVIER A. DIEZ, Universidad Nacional del Centro de la Provincia de Buenos Aires — We present a combined computational and asymptotic approach to study evolution of liquid rings on partially wetting solids. The work is motivated by recent experiments involving the self-assembly of nanolithographically patterned metal rings into ordered nanoparticle arrays via nanosecond pulsed laser heating above the melt threshold (Langmuir 2010, 26, 11972). Three-dimensional numerical simulations of Navies-Stokes equations are compared with a hydrodynamic model based on lubrication approximation for the dynamics and shape of liquid rings. Good agreement is found for contact angles smaller than approximately 35 degrees. The results are encouraging from two viewpoints: a simple hydrodynamic model can capture the main features of the processes under small slope conditions, and offer the possibilities of further developments of computational approaches for modeling fluid dynamics on nanoscale.

8:26AM G20.00003 Weighted integral boundary layer method for thin films along parallel undulating structures , MOHAMMED RIZWAN SADIQ IQBAL, Department of Mathematics and Statistics, University of Konstanz — A first order weighted integral boundary layer method is derived for a falling film along parallel undulated structures under isothermal conditions. The linear stability of the equilibrium flow, which is derived under asymptotic approach based on small amplitude undulations of the system, is assessed theoretically under certain limiting cases. The complex eigenvalue problem for the linear stability is studied numerically on a periodic domain. For the flow on a planar geometry, the linear stability results of the present model are found to be in good agreement with the experimental data available in literature and predicts the results at the Hopf bifurcation threshold accurately. For an undulating geometry, the amplitude of the undulations play a crucial role in stabilizing the film flow. The evolution triggered by small disturbances to the equilibrium profile is studied numerically by integrating the system using a fast Fourier transform to evaluate the spatial derivatives and a fourth order Runge-Kutta scheme to march in the temporal direction. The maximum film thickness profile computed numerically confirms the stabilizing nature of undulations of the structure as shown by the linear stability results.

8:39AM G20.00004 Numerical simulations of miscible channel flow with chemical reactions[1] , J.N. KUSUMA, KIRTI SAHU, Indian Institute of Technology, Hyderabad, OMAR MATAR, Imperial College London — We investigate the pressure-driven miscible displacement of one fluid by another in a horizontal channel in the presence of an exothermic chemical raction. We solve the continuity, Navier- Stokes, and energy conservation equations coupled to convective- diffusion equations of the reactant and product. The viscosity is assumed to depend on the volume fraction of the reactant and product as well as the temperature. The effects of relavent paramaters such as, Reynolds number, Peclet number, Damköhler number and the viscosity ratio of the reactant and product will be presented at the meeting.

[1]EPSRC Grant number EP/E046029/1

8:52AM G20.00005 Surface-tension-driven convection in pure liquid layers evaporating into ambient air: influence of liquid volatility[1] , FABIEN CHAUVET, SAM DEHAECK, PIERRE COLINET, ULB-TIPs, CP 165/67 — The spontaneous surface-tension-driven convective patterns induced by evaporation of a pure liquid layer are studied experimentally. A volatile liquid layer placed in a cylindrical container is left free to evaporate into air at rest under ambient conditions. The thermal dynamics of the evaporating liquid layer is visualized using an infrared camera. Evaporation rate and liquid thickness are measured by weighting. We focus on the transition between the convective state and the conductive state appearing at a certain instant during the drying of the liquid layer. The critical Marangoni number Ma_c associated to this transition is estimated from evaporation rate and layer thickness measurements at this instant. The effect of the evaporation rate on Ma_c and k_c (the critical wavenumber) has been investigated by changing the container height and, separately, the effect of the liquid volatility has been studied by using different liquids. Interestingly, it appears that Ma_c does not depend on the evaporation rate while it depends strongly on the liquid volatility. Given the typical uncertainties associated with liquid properties, a quite reasonable agreement is found with a "one-sided" linear stability analysis of this problem.

[1]Supported by ESA & BELSPO, by the EU, by ULB, and by FRS - FNRS.

9:05AM G20.00006 Dripping and jetting regimes in co-flowing capillary jets: unforced measurements and response to driving , CHARLES BAROUD, MARIA-LUISA CORDERO[1], LadHyX, Ecole Polytechnique, France, FRANCOIS GALLAIRE, EPFL, Switzerland — We study the breakup of drops in a co-flowing jet, within the confinement of a microfluidic channel. The breakup can occur right after the nozzle (dripping) or through the generation of a liquid jet that breaks up a long distance from the nozzle (jetting). Traditionally, these two regimes have been considered to reflect an absolutely unstable jet or a convectively unstable jet, respectively. We first provide measurements of the frequency of oscillation and breakup of the liquid jet; the dispersion relation thus obtained compares well with existing theories for convective instabilities in the case of the jetting regime. However, the theories in the absolutely unstable mode fail to predict the evolution of the frequency and drop size in the dripping regime. We also test the jet response to an external forcing, using a focused laser to locally heat the jet. The dripping regime is found to be insensitive to the perturbation and the frequency of drop formation remains unaltered. In contrast, the jetting regime locks to the external frequency, which translates into a modification of the drop size in agreement with the dispersion relations. This confirms the convective nature of the jetting regime.

[1]Permanent address: Universidad de Chile

9:18AM G20.00007 The mechanism for difference in effects of the change in viscosity by chemical reactions on viscous fingering depending on Damköhler number , SHOHEI IWATA, YUICHRO NAGATSU, YUSUKE KONDO, KENJI MATSUDA, YOSHIHITO KATO, YUTAKA TADA, Nagoya Institute of Technology, Japan — In our previous study (Nagatsu et al., *J. Fluid Mech* 2007), we experimentally showed that an increase in the viscosity of the more-viscous liquid by a chemical reaction at a very high Damköhler number, Da, (defined as the ratio between a characteristic time of fluid motion and that of a chemical reaction) made viscous fingering pattern more dense. In present study, we investigated the effects of the increased viscosity caused by chemical reaction at a moderate Da on the viscous fingering in a radial Hele-Shaw cell. We observed that present reaction decreased the area occupied by the fingering pattern around the injection hole. As a result, the fingering pattern became less dense following the reaction. These results indicate that the effects of the increased viscosity by the reactions on viscous fingering pattern are different depending on Da. We discuss the mechanism for the difference based on visualization of reacting flow field by means of pH indicators.

9:31AM G20.00008 On the evolution of a retracting straight liquid sheet edge: experimental study , ROUSLAN KRECHETNIKOV, HANS C. MAYER, University of California at Santa Barbara — The evolution of the initially straight edge of a retracting liquid sheet is still a subject of debate. Theoretical and numerical studies have provided conflicting results, and experimental efforts have, to our knowledge, never been attempted owing to the difficulty in achieving a uniform edge. However, recent advances in experimental techniques, specifically those presented in detail in Poster #72 of APS-DFD 2010 (H.C. Mayer and R. Krechetnikov), have allowed us to uniformly detach a soap film from a straight edge using an impulsively heated wire frame. The detachment, retraction, and breakup of soap films (h $\sim$ 10 μm) is analyzed using high speed photography. Owing to the Plateau border that connects the uniform film to the wire frames (wire diameter 25-250 μm) - a feature not present when rupturing films from a point - the early stages of retraction are dominated by a relatively large rim mass. We explore the phenomena at very early times (t $<$ 100 μs) associated with the birth of these detached films which may add complexity to the problem of determining what instability mechanism(s) are responsible for their breakup.

9:44AM G20.00009 Effects of gel properties produced by chemical reactions on viscous fingering , TOMOHIRO UJIIE, YUICHIRO NAGATSU, MITSUMASA BAN, SHUICHI IWATA, YOSHIHITO KATO, YUTAKA TADA, Nagoya Institute of Technology, Japan — We have experimentally investigated viscous fingering with chemical reaction producing gel. Here, two systems were employed. In one system, sodium polyacrylate (SPA) solution and ferric ion solution were used as the more and less viscous liquids, respectively. In another system, xthantan gum (XG) solution and the ferric ion solution were used as the more and less viscous liquids, respectively. For high concentration of ferric ion, viscous fingering pattern was changed into spiral pattern in the former system, whereas into fracture pattern in the latter system. We consider that the difference in the change of the patterns in the two systems will be caused by the difference in the properties of the gels. Therefore, we have measured the rheological properties of the gels by means of a rheometer. We have found that the gel in the former case is more elastic. Furthermore, we have discussed the relationship between the measured rheological properties and the observed spiral or fracturing patterns.

9:57AM G20.00010 Suppression of viscous fingering by elastic membranes , DRAGA PIHLER-PUZOVIC, University of Manchester, PIERRE ILLIEN, University of Manchester, ENS Paris, MATTHIAS HEIL, ANNE JUEL, University of Manchester — We report the suppression of interfacial fingering instabilities resulting from the injection of air into the narrow, fluid-filled gap between two parallel plates (a Hele- Shaw cell), when one of the boundaries is replaced by an elastic membrane. In a rigid cell, if the air is injected at a sufficiently fast flow rate, the interface is unstable, ultimately resulting in the development of a complex dendritic fingers. In the elastic- walled cell, interfacial instabilities develop at much larger air-injection rates because membrane deformations reduce destabilising pressure perturbations ahead of the propagating interface, and fundamentally alter the interfacial pattern that develops. We characterise the dependence of the instability on the parameters of the system, and present simple scaling arguments to explain the experimentally observed behaviour.

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G21 Vortex Dynamics III 324-325

8:00AM G21.00001 Evolving geometry of a vortex triangle , VIKAS KRISHNAMURTHY, HASSAN AREF, Engineering Science & Mechanics Department, Virginia Tech — We explore a new description of three-vortex motion in terms of the angles, A, B, C, in the triangle of vortices, the radius, R, of the circumcircle to this triangle, and the position of the circumcenter. We show that the equations of motion for the radius and angles may be expressed in terms of these variables alone, i.e., the evolution of R, A, B, C leads to an autonomous four-dimensional dynamical system. The evolution of the circumcenter, however, depends on the absolute vortex positions. Thus, the motion of three vortices may be regarded as a sum of two components: the motion of the circumcenter (giving the absolute position of the circumcircle) and, superimposed on this, the intrinsic dynamics of the size of the circumcircle and shape of the vortex triangle. Several results from the theory of three-vortex motion follow in this new representation and a number of new issues may be investigated.

8:13AM G21.00002 Bilinear relative equilibria of identical point vortices[1] , HASSAN AREF, Virginia Tech, PETER BEELEN, MORTEN BRØNS, Technical University of Denmark — A new class of bilinear relative equilibria of identical point vortices in which the vortices are constrained to be on two perpendicular lines, taken to be the x- and y-axes of a cartesian coordinate system, is introduced and studied. In general we have m vortices on the y-axis and n on the x- axis. We define generating polynomials $q(z)$ and $p(z)$, respectively, for each set of vortices. A second order, linear ODE for $p(z)$ given $q(z)$ is derived. Several results relating the general solution of the ODE to relative equilibrium configurations are established. Our strongest result, obtained using Sturm's comparison theorem, is that if $p(z)$ satisfies the ODE for a given $q(z)$ with its imaginary zeros symmetric relative to the x-axis, then it must have at least $n - m + 2$ simple, real zeros. For $m = 2$ this provides a complete characterization of all zeros, and we study this case in some detail. In particular, we show that given $q(z) = z^2 + \eta^2$, where η is real, there is a unique $p(z)$ of degree n, and a unique value of $\eta^2 = A_n$, such that the zeros of $q(z)$ and $p(z)$ form a relative equilibrium of $n + 2$ point vortices. We show that $A_n \approx \frac{2}{3}n + \frac{1}{2}$, as $n \to \infty$, where the coefficient of n is determined analytically, the next order term numerically.

[1]Supported in part by the Danish National Research Foundation through a Niels Bohr visiting professorship.

8:26AM G21.00003 ABSTRACT WITHDRAWN —

8:39AM G21.00004 New Results on Hollow Vortices[1] , STEFAN LLEWELLYN SMITH, UCSD, DARREN CROWDY, Imperial College — Hollow vortices are vortices whose interior is at rest. We obtain exact solutions for hollow vortices in linear and nonlinear strain. We then investigate the stability properties of these vortices and for the hollow vortex street of Baker, Saffman and Sheffield.

[1]Support from NSF CMMI-0970113

8:52AM G21.00005 Analysis of Reynolds number scaling for viscous vortex reconnection , QIONGLIN NI, Peking University, FAZLE HUSSAIN, University of Houston, JIANCHUN WANG, SHIYI CHEN, Peking University — A theoretical analysis of viscous vortex reconnection based on scale separation is developed, where the Reynolds number (Re = circulation/viscosity) scaling for reconnection time Trec is derived. The scaling varies from $T_{rec} \sim Re^{-1}$ to $T_{rec} \sim Re^{-0.5}$, and the direct numerical simulation (DNS) data from Garten [Garten et al, J. Fluid Mech. 426, 1 (2001), cited hereinafter as GW] and Hussain [Hussain et al, Phys. Fluids 23, 021701 (2011) cited hereinafter as HD] collapse well within the range of the asymptotic scalings. Moreover, our analysis predicts two Reynolds numbers, namely, a characteristic $Re_{theory} \in \left[O\left(10^2\right), O\left(10^3\right)\right]$ for the $T_{rec} \sim Re^{-0.75}$ scaling given by HD, and the critical Reynolds number $Re_c \sim O\left(10^4\right)$ for the transition after whom the large-scale vortex reconnection does no longer occur.

9:05AM G21.00006 Streamline Topology of Helical Fluid Flow , MORTEN ANDERSEN, Technical University of Denmark — An incompressible velocity field with helical symmetry is investigated. Helical symmetry means the velocity field "looks" the same as one moves on a given helix. This means a stream function can be constructed. The focus is on a proper description of helical symmetry and a bifurcation analysis of a flow with helical symmetry - the helical vortex filament.

9:18AM G21.00007 The constrained dipole dynamical system , ANDREW TCHIEU, EVA KANSO, PAUL NEWTON, University of Southern California — In this presentation the notion of a *constrained dipole* is introduced as a pair of equal and opposite strength point vortices (i.e. a vortex dipole) separated by a finite distance such that this distance remains constant throughout the time evolution of the vortex dipole. The current formulation unambiguously defines a self-propelling velocity for the dipole. Equations of motion for N constrained dipoles interacting in an unbounded inviscid fluid are derived from the modified interaction of $2N$ independent vortices subject to the constraint that the inter-vortex spacing of each constrained dipole remains constant. There are no far-field assumptions for the dipole-dipole interactions. We discuss the dynamics of the system of equations in the context of uncoordinated self-propelled motion in a perfect fluid and give examples of interactions of two, three, and many constrained dipoles. Dipoles are found to collide with one another in abreast formations. Alternatively, the dipoles are conducive to remaining in formation when placed in staggered and diamond formations. Interestingly, equilibria are also formed from these aforementioned interactions, and we investigate the polygonal configurations of constrained dipoles and their subsequent stability. It is found that the $N = 3$ case is linear stable, whereas $N > 3$ is linearly unstable.

9:31AM G21.00008 Coupling of a 3-D vortex particle-mesh method with a finite volume near-wall solver , Y. MARICHAL[1], Universite catholique de Louvain (UCL) - iMMC, T. LONFILS, Cenaero, M. DUPONCHEEL, P. CHATELAIN, G. WINCKELMANS, UCL - iMMC — This coupling aims at improving the computational efficiency of high Reynolds number bluff body flow simulations by using two complementary methods and exploiting their respective advantages in distinct parts of the domain. Vortex particle methods are particularly well suited for free vortical flows such as wakes or jets (the computational domain -with non zero vorticity- is then compact and dispersion errors are negligible). Finite volume methods, however, can handle boundary layers much more easily due to anisotropic mesh refinement. In the present approach, the vortex method is used in the whole domain (overlapping domain technique) but its solution is highly underresolved in the vicinity of the wall. It thus has to be corrected by the near-wall finite volume solution at each time step. Conversely, the vortex method provides the outer boundary conditions for the near-wall solver. A parallel multi-resolution vortex particle-mesh approach is used here along with an Immersed Boundary method in order to take the walls into account. The near-wall flow is solved by OpenFOAM® using the PISO algorithm. We validate the methodology on the flow past a sphere at a moderate Reynolds number.

[1]F.R.S. - FNRS Research Fellow

9:44AM G21.00009 Resonant instability in two-dimensional vortex arrays , PAOLO LUZZATTO-FEGIZ, Woods Hole Oceanographic Institution, CHARLES H.K. WILLIAMSON, Cornell University — In this work, we examine conditions for the development of an oscillatory instability in two-dimensional vortex arrays. By building on the theory of Krein signatures for Hamiltonian systems, and considering constraints owing to impulse conservation, we show that a resonant instability (developing through coalescence of two eigenvalues) cannot occur for one or two vortices. We illustrate this deduction by examining available linear stability results for one or two vortices. Our work indicates that a resonant instability may, however, occur for three or more vortices. For these more complex flows, we propose a simple model, based on an elliptical vortex representation, to detect the onset of an oscillatory instability. We provide an example in support of our theory by examining three co-rotating vortices, for which we also perform a linear stability analysis. The stability boundary in our model is in good agreement with the full stability calculation. In addition, we show that eigenmodes associated with an overall rotation or an overall displacement of the vortices always have eigenvalues equal to zero and $\pm i\Omega$, respectively, where Ω is the angular velocity of the array. These results, for overall rotation and displacement modes, can also be used to immediately check the accuracy of a detailed stability calculation.

9:57AM G21.00010 3D Characterization of Transmitral Vortex using Defocusing Digital Particle Image Velocimetry , AHMAD FALAHATPISHEH, BRANDON DUEITT, University of California Irvine, NIEMA PAHLEVAN, California Institute of Technology, ARASH KHERADVAR, University of California Irvine — In this study, we have experimentally characterized the 3D vortex passing through a physiologically relevant model of mitral valve using Defocusing Digital PIV (DDPIV). The valve model was made of soft silicone with diameter of $25mm$, similar to the adult mitral valve. The mitral model possesses a large anterior and a small posterior leaflet that results in asymmetric formation of transmitral vortex. A piston-cylinder mechanism drives the flow and travels to produce a range of L/D from 2 to 6. We have characterized the shape of the 3D vortex forming through the D-shaped orifice of a mitral valve using DDPIV technique. The evolution of the vortex has been illustrated for different stroke ratios.

Monday, November 21, 2011 8:00AM - 9:57AM –
Session G23 Separated Flows I 326

8:00AM G23.00001 Unsteady aerodynamic models for separated flow past a flat plate at Re=100 , STEVEN BRUNTON, CLARENCE ROWLEY, Princeton University — This work develops reduced-order models for the unsteady aerodynamic forces on a small wing in response to agile maneuvers and gusts. Models are based on direct numerical simulations performed using the immersed boundary projection method for a flat plate at Reynolds number 100. In particular, we investigate models linearized at various angles of attack, up to the Hopf bifurcation, which occurs around $\alpha_{\mathrm{crit}} = 28°$. A main result is that we present a flexible, low-order representation that includes viscous separated effects as well as added-mass effects, and may be seen as a linearization of a physically motivated nonlinear dynamical system. The performance of models below the Hopf bifurcation are investigated in the frequency domain as well as on large amplitude, rapid pitch maneuvers in the time domain. Additionally, we investigate models and flow field structures in the fully nonlinear vortex shedding regime using force, vorticity, and finite-time Lyapunov exponent field measurements.

8:13AM G23.00002 Anisotropy-resolving models for predicting separation in 3–D asymmetric diffusers , ELBERT JEYAPAUL, PAUL DURBIN[1], Iowa State University — All linear eddy-viscosity models are qualitatively incorrect in predicting separation in 3–D asymmetric diffusers. The failure to predict normal stress and shear stress anisotropy at high production-dissipation ratios is the cause. The Explicit algebraic Reynolds stress model (Wallin and Johansson, 2000) predicts the mean flow field in the diffuser accurately, but not the wall pressure and Reynolds stresses. Recalibrating the coefficients of the rapid part of pressure-strain model improves the wall pressure prediction. Including the convective, diffusive, streamline curvature effects on anisotropy has not been beneficial. The model has been tested using a family of diffusers having the same nominal streamwise pressure gradient, LES data is used as a reference.

[1]Professor

8:26AM G23.00003 A numerical investigation of the laminar, three-dimensional backward-facing step[1] , NIKOLAOS MALAMATARIS, TEI W. Macedonia / George Mason University, RAINALD LOHNER, George Mason University — The three-dimensional, backward-facing step is a benchmark problem in CFD. However, only recently, Malamataris and Löhner (2011) were able to compute the eddy along the upper wall of the step for this flow in laboratory feasible conditions, although the existence of this eddy is known from the experiments of Armaly et al (1983) and Lee and Mateescu (1998). There are some misinterpretations regarding three-dimensional effects for this flow due to the fact that at the time of the work of Armaly et al (1983) only two-dimensional computations were available that perpetuated until the work of Malamataris and Löhner. The issues with three-dimensional effects are shown to leave the flow phenomena unaffected in the plane of symmetry for the conditions of Armaly et al's experiment by showing streamlines in planes parallel/perpendicular to the plane of symmetry. The Reynolds number ranges from 100 to 1050. The steady state, three-dimensional Navier Stokes equations are solved with finite elements using two different codes FEFLO and FEM3D.

[1]Computational resources GMICE (George Mason University), allocation CTS 080031 (NSF) .

8:39AM G23.00004 Flow features from the wake of a rotating sphere , ANDREW OOI, ERIC POON, The University of Melbourne, MATTEO GIACOBELLO, Defence Science Technology Organisation, RAYMOND COHEN, CSIRO — The flow features in the wake of a rotating sphere is investigated numerically. Simulations are carried out at Reynolds numbers up to Re = 1000, with non-dimensional rotational rates Ω= 0 to 1. The axis of rotation is orientated from $\alpha = 0$ (streamwise rotation) to $\alpha = \pi/2$ (transverse rotation). At Re=250 and 300, the flow field could be steady or unsteady, depending on the value of Ω and α. Phase diagrams of (CLx,CLy) and (CD,CL) are used to illustrate the dynamics of the unsteady forces acting on a rotating sphere. By employing the phase diagram (CLx,CLy), the direction of the resultant lift force can be shown to be dependent on the non-dimensional rotation rate. The shape of the curves which appear on the phase diagrams (CLx,CLy) are highly correlated to the behaviour of the vortical structures in the wake. This study will give a more complete picture of the physics associated with the incompressible flow past a rotating sphere.

8:52AM G23.00005 A Nonlinear Model for Wind-Induced Oscillations of Open-Grown Trees , LAKSHMI NARAYANAN RAMANUJAM, MARK REILAND, YAHYA MODARRES-SADEGHI, BRIAN KANE — Wind-induced oscillations of trees constitute a traditional fluid-structure interaction problem. The highly nonlinear interaction between fluid and structure makes the prediction of such oscillations very challenging. We use a nonlinear analytical model to predict the wind-induced oscillations of the open-grown trees, by taking into account the geometrical as well as fluid-related nonlinearities. We validate this model using a series of experimental measurements of the amplitudes and frequencies of oscillations for the red pine trees at various wind speeds. Then we use this model to study the influence of several parameters such as tree stiffness, drag coefficient, canopy density, etc. on the resulting wind-induced oscillations.

9:05AM G23.00006 DNS study of large-scale structures in a separated turbulent boundary layer , HIROYUKI ABE, YASUHIRO MIZOBUCHI, YUICHI MATSUO, Japan Aerospace Exploration Agency — Direct numerical simulations (DNSs) of a separated flat-plate turbulent boundary layer have been carried out. The inlet data are prescribed by DNSs of a zero-pressure-gradient turbulent boundary layer with the rescaling-recycling method; blowing and suction are imposed at the upper boundary for producing a separation bubble. The Reynolds numbers at the inlet are set to be Re_θ=300, 600 and 900, where Re_θ is the Reynolds number based on the freestream velocity and the momentum thickness. Particular attention is given to large- scale structures existing in a separated region. Results indicate that large-scale organized structures of the streamwise velocity fluctuation appear in a detached shear layer when a large separated region is formed. The latter structures consist of positive and negative regions alternating in the spanwise direction with a spacing of about $2 \sim 3\delta_{99}$ (δ_{99} denotes the 99% boundary layer thickness at the inlet), which become more apparent with increasing Reynolds number. They are most likely associated with large-scale spanwise meandering of the separation line. There is also close relationship between the large-scale structures and vortical structures, the latter tending to form vortex clusters where hairpin-like vortices are also observed.

9:18AM G23.00007 Three–dimensional Instability in the Wake of Two Side–by–side Square Cylinders[1] , CHOON-BUM CHOI, Inha University, Korea, YONG-JUN JANG, Korea Railroad Research Institute, Korea, KYUNG-SOO YANG, Inha University, Korea — Three-dimensional instability in the flow past two square cylinders in side-by-side arrangements has been numerically studied via a Floquet stability analysis. The distance between the neighboring faces of the two cylinders (G) is the key parameter which affects the secondary instability under consideration. In this talk, we present the critical Reynolds number for the 3D instability and the spanwise wave number corresponding to the most unstable (or least stable) wave for each G. When the two cylinders are close to each other, the Floquet mode resembles mode A of flow past a single cylinder. For large G, the Floquet mode for each cylinder is independent of the neighboring cylinder as expected. In the range of intermediate G, interference of the two wakes is clearly reflected in the Floquet mode. We also discuss about the effect of gap flow, and visualization of the dominant 3D vortical structures is provided by using Q contours. Our results shed light on a complete understanding of the onset of 3D instability in the presence of two side-by-side square cylinders.

[1]This work was supported by Korea Railroad Research Institute (Research Title: The study of standardization for urban railway facilities (SW 11002)).

9:31AM G23.00008 Separation Bubble Dynamics in Single and Multiple STBLIs[1] , BRANDON MORGAN, NGHIA NGUYEN, SANJIVA LELE, Stanford University — We first analyze the low-frequency motion of incipient shock-induced separation using the results of large-eddy simulation (LES) of an oblique shock impinging on a supersonic turbulent boundary layer ($M_\infty = 2.28$, $\varphi = 8^\circ$, $\mathrm{Re}_\theta = 2300$). Through analysis of low-pass filtered data fields, evidence is found of a correlation between the low-frequency motion and a periodic exchange of mass between the separation bubble and shear layer. Integral mass and momentum budgets in filtered bubbles are computed from the LES data to gain insight into the mechanism of low-frequency unsteadiness, and time signals of several indicators of bubble dynamics are analyzed. Additionally, a preview of results is presented from LES of a normal shock train in a constant area duct. Significant three-dimensionality is observed, and its effects on separation dynamics are described, when viscous side-wall boundary conditions are imposed in the spanwise direction.

[1]This work has been supported by the NDSEG fellowship and the DoE PSAAP program. Computer time has been provided by NASA and LLNL.

9:44AM G23.00009 ABSTRACT WITHDRAWN –

Monday, November 21, 2011 8:00AM - 9:57AM –
Session G24 Rayleigh-Benard Convection IV: Turbulence 327

8:00AM G24.00001 ABSTRACT WITHDRAWN –

8:13AM G24.00002 Wind velocities and their connection to boundary layer characteristics in turbulent Rayleigh-Bénard convection[1] , SEBASTIAN WAGNER, OLGA SHISHKINA, CLAUS WAGNER, German Aerospace Center (DLR), Institute of Aerodynamics and Flow Technology, Goettingen — Highly resolved direct numerical simulations have been performed for turbulent Rayleigh-Bénard convection in a cylindrical container with aspect ratio unity and Prandtl number Pr=0.786. The Rayleigh numbers in the simulations are up to 10^9. The calculated fields are reduced to the plane of the large scale circulation and analyzed with respect to the viscous and thermal boundary layer thickness and corresponding quantities in a statistical manner, i.e. probability density functions and their characteristics have been calculated a posteriori from instantaneous flow fields. Different methods to determine the boundary layer thickness are suggested and compared. The results are analyzed and compared with the Prandtl-Blasius theory of laminar boundary layers as well as the Grossmann-Lohse theory of states in Rayleigh-Bénard convection. Thereby fundamental relations between wind velocities and the generated wall shear stress are determined.

[1]The authors acknowledge support by the Deutsche Forschungs Gemeinschaft (DFG) under grant SH405/2-1.

8:26AM G24.00003 Local fluctuations and fluxes in turbulent Rayleigh-Bénard convection , RAJARAM LAKKARAJU, RICHARD J.A.M. STEVENS, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, The Netherlands, ROBERTO VERZICCO, Department of Mechanical Engineering, University of Rome "Tor Vergata," Italy, SIEGFRIED GROSSMANN, Department of Physics, University of Marburg, Germany, CHAO SUN, DETLEF LOHSE, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, The Netherlands — We numerically investigate the local velocity and temperature fluctuations and the local heat flux in a cylindrical Rayleigh-Benard cell for Rayleigh numbers from Ra = 2 $\times 10^6$ to 2×10^9 at a fixed Prandtl number Pr = 5.2. The numerical measurements at different points in the cell reveal that the heat flux at the plume dominated edge is larger and that it always has a power law relationship with *Ra*, i.e. $j \sim A\ Ra^{\gamma}$. The scaling exponent linearly depends on the radial distance from the center, with = 0.44 at the center to = 0.27 at the edge, similarly has found by Shang, Tong and Xia, PRL 2008 and predicted by Grossmann and Lohse, Phys. Fluids 2004. These values imply a transition to bulk dominated heat flux beyond Ra = 10^{15}. The velocity and temperature fluctuations also have a power law dependence on *Ra* and the corresponding scaling exponents again dependent on the radial position.

8:39AM G24.00004 Studies of the boundary layer structure in turbulent Rayleigh-Bénard convection[1] , JOERG SCHUMACHER, NAN SHI, MOHAMMAD EMRAN, TU Ilmenau, D-98684 Ilmenau, Germany — The structure of the laminar boundary layer in turbulent Rayleigh-Bénard convection is studied by three-dimensional direct numerical simulations. We consider convection in a cylindrical cell at an aspect ratio of one for Rayleigh numbers of 3×10^9 and 3×10^{10} and at a Prandtl number of $Pr = 0.7$. The laminar boundary layers of the velocity and temperature fields are found to deviate from the classical prediction of the Prandtl-Blasius-Pohlhausen theory, even when a dynamical rescaling of the data with an instantaneously defined thickness scale is performed in a plane that is aligned with the instantaneous orientation of the large-scale wind. We show on the basis of the existing numerical data that none of the assumptions that enter this classical solution are satisfied. Three-dimensional structures are present and important, the large-scale wind changes direction and amplitude and the pressure gradients are found to fluctuate significantly. The local boundary layer structure is compared with perturbative expansions of the boundary layer equations of forced and natural convection.

[1]This work is supported by the Deutsche Forschungsgemeinschaft within FOR 1182.

8:52AM G24.00005 Transport Velocities in the Convection Zone of the Sun Are Anomalously Weak[1] , KATEPALLI SREENIVASAN, New York University, SHRAVAN HANASOGE, Princeton University, THOMAS DUVALL, JR., NASA Goddard Space Flight Center — Understanding the dynamo action in the Sun, the process by which its magnetic fields are created and sustained, remains a major challenge. Our current understanding of interior convective turbulence, thought to play a critical role in regulating the dynamo, is derived solely from phenomenology and simulation, neither of which addresses the appropriate parameter regime. Here, we seismically constrain transport velocity magnitudes as a function of spherical-harmonic degree ℓ and depth in the convection zone to be 20 - 100 times lower than predictions from the current simulation methodologies and mixing length models. These bounds are obtained by isolating waves that propagate to specific depths and searching for statistical evidence for deviations in their travel times due to advection by convective flows. Turbulence on scales $\ell \leq 60$ persists in a strongly rotationally-dominated regime of *geostrophy* since the Rossby numbers at $r/R_\odot = 0.96$, where $R_\odot$ is the solar radius, are less than $\sim 10^{-2}$.

[1]Computing was done on NASA Ames machines, Schirra and Pleiades.

9:05AM G24.00006 On Unsteady Natural Convection Between Spherical Shells[1] , YURI FELDMAN, TIM COLONIUS, Division of Engineering and Applied Science, California Institute of Technology — Natural convection between two concentric spheres is investigated with three-dimensional numerical simulations. Buoyancy is achieved by preserving a temperature difference between the internal hotter and the external colder boundaries of the spherical shell. The numerical simulations were performed for the two basic configurations characterized by external to internal radius ratios of 1.2 and 1.5. Slightly supercritical laminar regimes characterized by the Rayleigh numbers of order Ra $\sim O(10^4\text{-}10^5)$ were simulated by utilizing a Direct Numerical Simulation (DNS) approach while a Large Eddy Simulation (LES) was used for investigation of turbulent regimes for Ra $\sim O(10^8\text{-}10^9)$. We discuss the topological characteristics of the both laminar and turbulent flows. One of the possible scenarios of steady-unsteady transition is proposed as well. Implications of the results for the design of a double-walled Montgolfiere aerobot for the exploration of Titan's atmosphere are discussed.

[1]Research supported by Jet Propulsion Laboratory with Dr. Jeffrey Hall as monitor.

9:18AM G24.00007 Turbulent Rayleigh-Bénard convection of a nematic liquid crystal[1] , STEPHAN WEISS, GUENTER AHLERS, University of California, Santa Barbara — While studying Rayleigh-Bénard convection (RBC) of Newtonian fluids is a challenging and interesting topic on its own, we want to expand this field by including anisotropic fluids such as liquid crystals. In these liquids certain fluid properties, such as for example the heat conductivity λ and the diamagnetic susceptibility χ, depend on the average orientation of the molecules (the director). We present results from experiments where the nematic liquid crystal 4-n-pentyl-4'-cyanobiphenyl (5CB) was used as a working fluid in turbulent RBC. Since χ is anisotropic, one can alter the director orientation, and thus λ, by applying a magnetic field H. Our measurements show that the effective heat conductivity $\lambda_{eff} = q/\Delta T$ of the turbulent sample increases quadratically with H (ΔT is the temperature difference between top and bottom plate and q the corresponding heat flux). We suggest a simple model that considers the interplay between the director-aligning magnetic field and the vigorous fluid motion that randomize the director orientations inside the thermal boundary layers at the top and bottom plates.

[1]This work was supported by NSF Grant DMR07-02111.

9:31AM G24.00008 Enhanced and Reduced Heat Transport in Turbulent Thermal Convection with Polymer Additives[1] , PING WEI, RUI NI, KE-QING XIA, Department of Physics, The Chinese University of Hong Kong — We present an experimental study of heat transport in turbulent Rayleigh-Bénard (RB) convection with polymer additives in convection cells both with smooth and rough surfaces on the top and bottom solid conductive plates. For the cell with smooth plates, a reduction of the measured Nusselt number (Nu) was observed. Furthermore, the amount of Nu- reduction increases with increasing polymer concentration (c), reaching $\sim 12\%$ for $c = 120$, ppm and an apparent saturation of the Nu-reduction thereafter. For the cell with rough plates, however, an enhancement of Nu of was observed when the polymer concentration is greater than 120.

[1]This work was supported by RGC of Hong Kong SAR (No. CUHK403807 and 404409).

9:44AM G24.00009 Connecting flow structures and heat flux in turbulent Rayleigh-Bénard convection , ERWIN VAN DER POEL, RICHARD STEVENS, DETLEF LOHSE, University of Twente — The aspect ratio (Γ) dependence of the heat transfer (Nusselt number Nu in dimensionless form) in turbulent (two- dimensional) Rayleigh-Bénard convection is numerically studied in the regime $0.4 \leq \Gamma \leq 1.25$ for Rayleigh numbers $10^7 \leq Ra \leq Ra^9$ and Prandtl numbers $Pr = 0.7$ (gas) and 4.3 (water). $Nu(\Gamma)$ shows a very rich structure with sudden jumps and sharp transitions. We connect these structures to the way the flow organizes itself in the sample and explain why the aspect ratio dependence of Nu is more pronounced for small Pr. Even for fixed Γ different turbulent states (with different resulting Nu) can exist, between which the flow can or cannot switch. In the latter case the heat transfer thus depends on the initial conditions.

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G25 Suspensions II 328

8:00AM G25.00001 Osmotic motor under fixed flux conditions , UBALDO CORDOVA-FIGUEROA, University of Puerto Rico, JOHN BRADY, California Institute of Technology, SERGEY SHKLYAEV, University of Puerto Rico — We consider an osmotic motor that emits product particles from half of its surface. This asymmetry produces a difference of the product concentration, and hence of the osmotic pressure, between the active and passive surfaces, resulting in motion of the motor. Hydrodynamic interactions between the motor and the small product particles are included in the analysis following the theory developed by Brady (JFM, 2011). Limiting cases of slow and fast motion of the motor are considered analytically; otherwise the velocity of self-propulsion is determined numerically. Brownian Dynamic simulations are in a good agreement with the theoretical predictions. The analysis is also generalized to a reactive patch, which is smaller or larger than a hemisphere; in either case this diminishes the motor velocity.

8:13AM G25.00002 Chemical sailing: Nonspherical catalytic motors , SERGEY SHKLYAEV, University of Puerto Rico, JOHN BRADY, California Institute of Technology, UBALDO CORDOVA-FIGUEROA, University of Puerto Rico, GROUP ON COLLOIDAL DYNAMICS TEAM — Self-propulsion of a chemically active particle (e.g. an osmotic motor) is a promising area that has a number of far- reaching applications. The conventional strategy is to construct a Janus particle that is chemically active on a portion of its surface, whereby an asymmetric concentration gradient of reactant is generated leading to osmotic propulsion. However, a nonspherical motor with uniform intensity of chemical reaction on its surface can also lead to osmotic propulsion, which allows for the design of simpler, cheaper motors. Wei & Jan (JFM, 2010) considered a weakly nonspherical motor and found the velocity of self-propulsion to be linear in terms of nonsphericity. However, we show that their calculations are in error and that net propulsion occurs at second order with respect to the nonsphericity. Moreover, hydrodynamic interactions (HI), which were neglected by Wei & Jan, can be of crucial importance—even the direction of self- propulsion can change with HI. Large departures from sphericity are also investigated numerically in an effort to determine the optimum shape for maximum propulsion.

8:26AM G25.00003 A liquid bridge connecting moving porous surfaces[1] , MORTEZA GHARIB, AMIR GAT, Caltech, HOMAYUN NAVAZ, Kettering University — We study the coupled problem of a liquid bridge connecting two porous surfaces where the gap between the surfaces is an externally controlled function of time. The relative motion between the surfaces affects the geometry and the pressure distribution of the liquid bridge, thus influencing the diffusion speed and penetration topology within the porous material. Utilizing the lubrication approximation and Darcy's phenomenological law we obtain a relation between the diffusion into the porous surface and the relative motion between the surfaces. A scheme to control the diffusion topology is presented and illustrated for the case of conical penetration topology with an arbitrary cone opening angle. Analytic expressions describing the penetration topology for the case of constant speed of the surfaces and the relative motion between the surfaces required to create a conical penetration topology are obtained and compared to experimental and numerical data.

[1]This project was supported by the Defense Threat Reduction Agency (DTRA), Award Number: 330233-A.

8:39AM G25.00004 Hydrodynamic interactions between two vesicles , PIERRE-YVES GIRES, Grenoble University, DYFCOM TEAM — A giant vesicle is a closed elastic membrane containing a liquid, inside another liquid. Its size is around 10 microns. If a suspension of such objects is sheared, they sometimes come close and interact hydrodynamically. We studied how these interactions affect the trajectories of the vesicles. For this, we model the properties of the membrane, assuming that the area of a surface element is constant in the course of time, and that it resists bending. We also assume that the inside and outside fluids are Newtonian, and are in the creeping regime. To solve the partial differential equations arising from this model, we used two methods : an asymptotic expansion around spherical shapes for vesicles far away from each other (3d case), and a boudary integral method (2d case). We find that vesicles repel, and that this repulsion decreases with initial transverse distance. We compare our results with experimental results performed with vesicles flowing in microfluidic devices.

8:52AM G25.00005 Convective Polymer Depletion on Pair Particle Interactions[1] , TAI-HSI FAN, Department of Mechanical Engineering, University of Connecticut, USA, TAKASHI TANIGUCHI, Department of Chemical Engineering, Kyoto University, Japan, REMCO TUINIER, DSM Research, The Netherlands — Understanding transport, reaction, aggregation, and viscoelastic properties of colloid-polymer mixture is of great importance in food, biomedical, and pharmaceutical sciences. In non-adsorbing polymer solutions, colloidal particles tend to aggregate due to the depletion-induced osmotic or entropic force. Our early development for the relative mobility of pair particles assumed that polymer reorganization around the particles is much faster than particle's diffusive time, so that the coupling of diffusive and convective effects can be neglected. Here we present a nonequilibrium two-fluid (polymer and solvent) model to resolve the convective depletion effect. The theoretical framework is based on ground state approximation and accounts for the coupling of fluid flow and polymer transport to better describe pair particle interactions. The momentum and polymer transport, chemical potential, and local viscosity and osmotic pressure are simultaneously solved by numerical approximation. This investigation is essential for predicting the demixing kinetics in the pairwise regime for colloid-polymer mixtures.

[1]This work is supported by NSF CMMI 0952646.

9:05AM G25.00006 Non-equilibrium depletion interactions: dual-probe microrheology , ROSEANNA ZIA, JOHN BRADY, California Institute of Technology — Non-equilibrium depletion interactions in colloidal dispersions are studied via non-linear, dual-probe microrheology, theoretically and via Brownian dynamics simulation. We study the interactive force between a pair of probe particles translating with equal velocity through a colloidal dispersion with their line of centers transverse to the external forcing. The character of the microstructure surrounding the probes is determined by the distance R by which the two probes are separated and by the strength of the external forcing compared to the thermal force of the bath, which defines a Péclet number, $Pe = Ua/D_b$, where U is the probe velocity, a is its size and D_b the diffusivity of the bath particles. Osmotic pressure gradients develop as the microstructure is deformed, giving rise to an interactive force between the probes. This force is studied for a range of Pe and R. For all separations $R > 2a$, the probes attract when Pe is small. As the strength of the forcing increases, a qualitative change in the interactive force occurs: the probes repel each other. The separation R at which the attraction-to-repulsion transition occurs decreases as Pe increases as the entropic depletion attraction becomes weak compared to the force-induced osmotic repulsion.

9:18AM G25.00007 Lattice Boltzmann Method For Structured Flexible Bodies in Fluid Flows , D. QI, Western Michigan University, T. WU, R. GUO, National Kaohsiung Normal University — A new simulation method for a structured flexible solid particles is developed. In this method, a spring lattice model is generalized to include two- and three-body force interactions and construct a structured fiber so that the fiber could be freely deformed in any irregular shapes. Lattice Boltzmann method and Drace Delta function are employed to deal with complex interaction in the interfaces between fluid and solid particles. In this new method, constrain forces are added into the fluid particles within the solid regime so that the inside fluid particles have the same property as the solid. Further, rotation and motion of a flexible fiber in a shearing flow and in a sediment flow, respectively, are numerically simulated.

9:31AM G25.00008 The effect of dissolved oxygen on electrohydrodynamic aggregation of colloidal particles near electrodes , T.J. WOEHL, N.D. BROWNING, W.D. RISTENPART, University of California at Davis — Colloidal particles suspended in dilute electrolytes have been widely observed to aggregate laterally along electrodes in response to oscillatory electric fields. Studies on the effects of particle type and electrode material have suggested that electrochemical reactions play a role in driving aggregation, but the detailed mechanism remains unclear. Here we demonstrate that, at sufficiently low frequencies, the dissolved oxygen (DO) content of the suspending electrolyte strongly affects the aggregation behavior of micron-scale particles. At 100 Hz, colloids in aqueous KCl saturated with oxygen exhibit an aggregation rate three times larger compared to aggregation in deoxygenated solutions. In contrast, no effect of DO content is observed at 500 Hz with KCl, and no effect is observed at either frequency when the suspending electrolyte is KOH. We investigate the role of DO using cyclic voltammetry, and we interpret the observations in terms of the effect of DO on the magnitude of the electrochemical current driving the electrohydrodynamic flow.

9:44AM G25.00009 Evolution of streamers in sedimentation of fibre suspensions bounded by vertical walls[1] , FENG ZHANG, ANDERS DAHLKILD, FREDRIK LUNDELL, Linne Flow Centre, KTH Mechanics, Royal Institute of Technology — The simulation, based on the Navier-Stokes equations coupled to a transport equation for the PDF of fibres, shows that a series of alternating structures of streamers and backflow regions emerge continuously from the walls until they converge in the midst of the domain. For moderate times, this agrees qualitatively with experimental and theoretical literature. In experimental literature, the evolution of the flow structure is restricted in time due to the finite height of the vessel and a steady state is not reached. However, our simulation in a vessel of infinite height obtained an increasing wavelength evolution due to the congregation of the streamers. In the end, there is constantly only one streamer left, and it drifts randomly to one side of the container until the evolution reaches a steady state. It also shows that the maximum number of streamers increases with increasing values of Re, fibres concentration, fibres aspect ratio, and container width.

[1]This work was supported by the Swedish Research Council (VR).

9:57AM G25.00010 Rheology of a dense capsule suspension[1] , R. MURTHY KALLURI, PROSENJIT BAGCHI, Rutgers University — The rheology of a dense suspension of deformable capsules in a linear shear flow is studied at low Reynolds numbers. Three- dimensional numerical simulations are conducted for initially spherical capsules using a front-tracking method. Capsules are represented as drops of Newtonian fluid enclosed by an elastic membrane and suspended in another Newtonian liquid of different viscosity. The capsule volume fraction ranges up to about 26%. This study is motivated by our earlier works [Bagchi & Kalluri, Phys Rev E, **81**, 056320 (2010); Bagchi & Kalluri, J Fluid Mech, **669**, 498 (2011)] where we show that for a *dilute* suspension the shear viscosity exhibits a nonmonotonic trend with respect to the internal-to-external fluid viscosity ratio; in particular, the shear viscosity exhibits a minimum at an intermediate viscosity ratio. In case of a *dense* suspension, we find that the shear viscosity minimum gradually diminishes as with increasing capsule volume fraction. We explain this result by decomposing the particle shear stress into elastic and viscous components. The elastic component is observed to increase but the viscous component remains constant with respect to increasing volume fraction. It is also shown that the elastic contribution is shear-thinning, but the viscous contribution is shear-thickening.

[1]Supported by NSF

Monday, November 21, 2011 8:00AM - 10:10AM – Session G26 Minisymposium: Electrokinetic Flows about Ion-Selective Surfaces 329

8:00AM G26.00001 Phoretic motion of ion exchangers , EHUD YARIV, Technion - Israel Institute of Technology — One-dimensional ionic conduction normal to an ion-exchange surface is a two-scale problem, with a charged Debye layer adjacent to the surface and an electro-neutral bulk outside it. Due to ion selectivity, the bulk salt concentration must vary linearly with distance. When the ion-exchange surface is curved, the electric field tangential to the surface results in an accompanying flow. Specifically, a freely suspended ion- exchange particle would move under the action of an external field. This problem is described in a systematic microscale model, wherein the boundary conditions on the highly-conducting particle surface represent ion-exchange kinetics. For thin Debye layers, an equivalent macroscale model is extracted, wherein boundary conditions reflect asymptotic matching with the Debye layer. Thus, the equi-potential condition on the literal particle surface is transformed to a nonuniform Dirichlet condition, accounting for Debye layer voltage (zeta potential). The latter is not a surface property, but rather depends upon surface kinetics. As in the one-dimensional problem, microscale ion selectivity on the particle surface results in a macroscale salt concentration polarization (on the particle scale), whereby the electric potential is rendered non-harmonic. Accordingly, the macroscale model is significantly more complicated than the comparable one for inert particles. Even in the weak-field limit, the effects of salt polarization and electric field on particle motion are comparable: Smoluchowski's formula does not apply. Open problems in the context of strong fields and self-propulsion will also be described.

8:26AM G26.00002 Deionization shocks in cross flows[1] , ALI MANI, Chemical Engineering, MIT, Cambridge MA, U.S.A.

— Recent experimental and theoretical studies have shown that surface conduction in supported electrolytes, such as in micro/nanochannels or porous media, can lead to nonlinear modes of transport and formation of sharp concentration fronts analogous to shock waves in gas dynamics. Propagation of these shocks leaves behind a region of ultra pure fluid, acting to deionize the bulk solution. In this work we present the analysis of salt transport in a porous medium next to a membrane with an electric field applied normal to the interface and cross flow in tangential direction. We show that two distinct boundary layers grow near the membrane: an inner (shocked) region with almost deionized solution dominated by surface conduction, and an outer layer with diffuse dynamics. Under certain conditions both regions collapse into a similarity solution with the same scaling. We will discuss advantages of such systems for desalination and water purification.

[1]Research performed in collaboration with Martin Bazant (MIT)

8:52AM G26.00003 Rectification Phenomena Across an Asymmetric Nanofluidic Channel ,

HSUEH-CHIA CHANG, Department of Chemical and Biomolecular Engineering, University of Notre Dame, IN — We review our recent experimental and theoretical studies of nanofluidic diodes. A continuum theory is developed to show that the literature and our rectification data can be generically classified into two regimes: a low-voltage regime dominated by intra- channel ionic strength (Donnan potential) gradient and a high-voltage regime dominated by external ion depletion. The two regimes drive different anomalous phenomena, like molecular dissociation and microvortex instability, with distinct distinguished limits of dimensionless parameters. Applications to biosensing are discussed.

9:18AM G26.00004 Concentration Polarization and Nonequilibrium Electro-osmotic Instability at an Ion-Selective Surface Admitting Normal Flow , ADITYA KHAIR, Chemical Engineering Department, Carnegie

Mellon University, Pittsburgh PA — We revisit and build upon the prototypical problem of ion transport across a flat ideal ion-selective surface. Specifically, we examine the influence of imposed fluid flows on concentration polarization (CP) and electrokinetic instability at over-limiting currents. We consider an ion-selective surface, or membrane, that admits a uniform flow across itself. The membrane contacts an electrolyte, whose concentration is uniform in a well-mixed region at a prescribed distance from the membrane. A voltage across the system drives an ionic current, leading to CP in the "unstirred layer" between the membrane and well-mixed bulk. The CP reflects a balance between advection of ions with the "normal flow" and diffusion. A Peclet number, Pe, parameterizes their relative importance; note, Pe is signed, as the flow can be toward or away from the membrane. An asymptotic analysis for thin Debye layers reveals a nonlinear CP profile, in contrast to the familiar linear profile at Pe=0. Next, we consider over-limiting currents, wherein a non-equilibrium space-charge layer emerges near the membrane surface. Finally, we examine the instability of the quiescent concentration polarization due to second-kind electro-osmosis in the space-charge layer. A stability analysis shows that the imposed normal flow can enhance or retard the instability, depending on its direction.

9:44AM G26.00005 Electrokinetics over liquid/liquid interfaces[1] , TODD M. SQUIRES, Chemical Engineering

Department, University of California, Santa Barbara — Since liquid-liquid interfaces flow in response to an applied stress, one might expect electrokinetic flows at liquid-liquid interfaces to be significantly higher than over liquid-solid interfaces. The earliest predictions for the electrophoretic mobility of charged mercury drops – distinct approaches by Frumkin and Levich (1946), and Booth (1951) – differed by $\mathcal{O}(a/\lambda_D)$, where a is the radius of the drop and λ_D is the Debye screening length. Seeking to reconcile this rather striking discrepancy, Levine (1973) showed double-layer polarization to be the key ingredient. Without a physical mechanism by which electrokinetic effects are enhanced, however, it is difficult to know how general the enhancement is – whether it holds only for liquid metal surfaces, or more generally, for all liquid/liquid surfaces. By considering a series of systems in which a planar metal strip is coated with either a liquid metal or liquid dielectric, we show that the central physical mechanism behind the enhancement predicted by Frumkin and Levich (1946) is the presence of an unmatched electrical stress upon the electrolyte-liquid interface, which establishes a Marangoni stress on the droplet surface and drives it into motion. The source of the unbalanced electrokinetic stress on a liquid metal surface is clear – metals represent equipotential surfaces, so no field exists to drive an equal and opposite force on the surface charge. This might suggest that liquid metals represent a unique system, since dielectric liquids can support finite electric fields, which might be expected to exert an electrical stress on the surface charge that balances the electric stress. We demonstrate, however, that electrical and osmotic stresses on relaxed double-layers internal to dielectric liquids precisely cancel, so that internal electrokinetic stresses generally vanish in closed, ideally polarizable liquids. The enhancement for liquid mercury drops can thus be expected quite generally over clean, ideally polarizable liquid drops. More broadly, the ability to reliably engineer liquid interfaces in microfluidic systems, then, may provide a path to significantly enhanced electrokinetic flows.

[1]Research done in collaboration with A. J. Pascall

Monday, November 21, 2011 8:00AM - 9:57AM –

Session G27 Focus Session: Wind Energy Fluid Dynamics III Ballroom I

8:00AM G27.00001 Atmospheric turbulence and its relevance for wind energy related research

, MICHAEL HÖLLING, ForWind, Institute of Physics - University of Oldenburg, ALLAN MORALES, MATTHIAS WÄCHTER, JOACHIM PEINKE — Positioned in the highly turbulent atmospheric boundary layer, wind turbines experience extreme wind conditions. By using the mean wind speed and the turbulence intensity on a ten minute basis to describe these turbulent wind fields, information about the chronology of the wind time series gets averaged out. Detailed knowledge about the wind field's behavior in time and their statistics can e.g. help to better estimate wind induced loads on wind turbines. We present a method to estimate the statistics of the fluctuations and more importantly the extreme fluctuations based on the ten minute averaged values of the turbulent intensity. In addition we present the effect of turbulent wind velocities on the lift coefficient of a FX 79-W-151A airfoil. Square grids and a fractal grid have been used to create turbulent flows with different turbulence intensities and increment statistics. Even though the turbulent intensity of the wind field generated by the fractal grid is smaller than of the wind field generated by the square grid, the results show a higher standard deviation in the measured lift coefficients. These increased fluctuations are a direct result from the generated atmospheric-like wind field with intermittent increment statistics on a wider range of scales created by the fractal grid.

8:13AM G27.00002 Effect of the Ekman Layer on the Effective Roughness Length of Large Scale Wind Farms , JAY PRAKASH GOIT, JORIS CODDÉ, HANS ROBEERS, JOHAN MEYERS, Katholieke Universiteit Leuven — Large scale

wind farms induce additional surface drag increasing the effective surface roughness ($z_{0,hi}$) experienced by upper atmospheric boundary layer (ABL). In a previous study, Calaf *et al.* [1] obtain $z_{0,hi}$ for wind farms when they are situated within the inner layer of the boundary layer, such that outer-layer dynamics, e.g., the effect of Coriolis forces induced by the Earth's rotation, may be neglected. However, for shallow ABLs where the wind-farm is not entirely situated in the inner layer, the effect of Coriolis forces may become important. In the present work, Large Eddy Simulations (LES) are used to addresses the effect of Coriolis forces on the evaluation of $z_{0,hi}$ for a very large wind farm. Results are compared with existing models for $z_{0,hi}$ [1, 2]. We find that the models for $z_{0,hi}$ agree well with the roughness obtained from the LES when wind turbines are well inside the inner region of the ABL. However, when the turbine height is more than 20% of the ABL, the model's predicted roughness height gets less accurate. This difference can be attributed to the influence of Coriolis Forces .

[1] Calaf M. *et al.* Phys Fluids, **22**, 015110, 2010.
[2] Frandsen S. *et al.* Wind Energy, **9**, 39-53, 2006.

8:26AM G27.00003 Statistical unfolding of atmospheric turbulence , ALLAN MORALES, PATRICK MILLAN, JOACHIM PEINKE, ForWind-University of Oldenburg, TWIST-FORWIND TEAM — Intermittent statistics, higher probability of extreme events in comparison to Gaussian statistics, is one of the trademarks of homogeneous isotropic turbulence (HIT). In the atmosphere, wind speed increments' probability density functions (PDFs) remain intermittent for a broader range of temporal and spatial scales. Moreover, wind speed fluctuations from the mean are intermittent for wind time series, which is not the case for HIT. In this work we revisit some features behind HIT with a free stream experiment and compare the results with those coming from atmospheric time series. We validate the idea that, within the inertial range, intermittency is coming from the randomness of the energy transfer rate. With a formulation, due to Castaing, based on Kolmogorov 62 (K62) we model increments' PDFs for packages of HIT in the atmosphere. With this knowledge and the observation that in the atmosphere not only the mean wind speed is non-stationary but also the turbulent kinetic energy, we are able to clearly identify two sources of intermittency in wind. We build a general model for modeling atmospheric turbulence statistics. Our model encapsulates mesoscale fluctuations, whereas high frequency turbulence can be treated and modeled with the full machinery developed in laboratory turbulence, in our case K62. We elaborate in the implications and utility of this work for Wind Energy.

8:39AM G27.00004 Development of a Wind Turbine Array Boundary Layer Under Thermally Stratified Conditions , ELIZABETH CAMP, ZACHARY WILSON, DOMINIC DELUCIA, RAÚL BAYOÁN CAL, Portland State University — Efforts have intensified in studying wind energy from a fluid mechanics and turbulence standpoint. Here, a wind turbine boundary layer is studied experimentally under stratified conditions. In this wind tunnel experiment, the mean velocities and turbulent quantities within a 3 by 3 scale-model wind turbine array are investigated. Cases for unstable and neutral boundary layer flows are described. These flows are modified upstream of the turbine array in order to emulate the atmospheric turbulent boundary layer using an active grid, strakes, thermally controlled floor panels, and roughness elements. All reported measured quantities are obtained through dual simultaneous stereo Particle Image Velocimetry systems which are made along the entire streamwise length of the array in order to chart the development of the flow.

8:52AM G27.00005 Identification of Flow Structures in a Stratified Wind Turbine Array Boundary Layer , MATT MELIUS, ZACHARY WILSON, ELIZABETH CAMP, RAÚL BAYOÁN CAL, Portland State University — Turbulent structures contained within a thermally stratified flow field as they convect through a wind turbine array are identified using instantaneous velocity fields obtained *via* particle image velocimetry (PIV). The experiment is conducted by placing a 3 by 3 scaled model wind turbine array in the test section of a wind tunnel. Using an active grid, strakes, and a thermally controlled tunnel floor, the conditions of a stratified atmospheric turbulent boundary layer are reproduced. Neutral and unstable conditions are compared. Vortical structures are captured both upstream and in the wake of the center turbine in the last row. The behaviors of the structures and how these evolve downstream are analyzed and compared to statistical quantities of the flow. Understanding these structures can prove important in determining the overall behavior of the flow and impact of the wind turbine array on the local environment.

9:05AM G27.00006 Influences of Atmospheric Stability State on Wind Turbine Aerodynamic Loadings , GANESH VIJAYAKUMAR, ADAM LAVELY, JAMES BRASSEUR, ERIC PATERSON, MICHAEL KINZEL, Penn State University — Wind turbine power and loadings are influenced by the structure of atmospheric turbulence and thus on the stability state of the atmosphere. Statistical differences in loadings with atmospheric stability could impact controls, blade design, etc. Large-eddy simulation (LES) of the neutral and moderately convective atmospheric boundary layer (NBL, MCBL) are used as inflow to the NREL FAST advanced blade-element momentum theory code to predict wind turbine rotor power, sectional lift and drag, blade bending moments and shaft torque. Using horizontal homogeneity, we combine time and ensemble averages to obtain converged statistics equivalent to "infinite" time averages over a single turbine. The MCBL required longer effective time periods to obtain converged statistics than the NBL. Variances and correlation coefficients among wind velocities, turbine power and blade loadings were higher in the MCBL than the NBL. We conclude that the stability state of the ABL strongly influences wind turbine performance. Supported by NSF and DOE.

9:18AM G27.00007 Inherent Variability in Short-time Wind Turbine Statistics from Turbulence Structure in the Atmospheric Surface Layer , ADAM LAVELY, GANESH VIJAYAKUMAR, JAMES BRASSEUR, ERIC PATERSON, MICHAEL KINZEL, Penn State University — Using large-eddy simulation (LES) of the neutral and moderately convective atmospheric boundary layers (NBL, MCBL), we analyze the impact of coherent turbulence structure of the atmospheric surface layer on the short-time statistics that are commonly collected from wind turbines. The incoming winds are conditionally sampled with a filtering and thresholding algorithm into high/low horizontal and vertical velocity fluctuation coherent events. The time scales of these events are ~5 - 20 blade rotations and are roughly twice as long in the MCBL as the NBL. Horizontal velocity events are associated with greater variability in rotor power, lift and blade-bending moment than vertical velocity events. The variability in the industry standard 10 minute average for rotor power, sectional lift and wind velocity had a standard deviation of ~ 5% relative to the "infinite time" statistics for the NBL and ~10% for the MCBL. We conclude that turbulence structure associated with atmospheric stability state contributes considerable, quantifiable, variability to wind turbine statistics. Supported by NSF and DOE.

9:31AM G27.00008 Spatial and Temporal Scale Dependence of Atmospheric Boundary Layer Turbulence , CHERYL KLIPP, US Army Research Laboratory — Turbulence affects wind turbine performance, often in ways that are not well understood. A better understanding of the atmospheric turbulence may help in understanding effects on the turbines. Analysis of atmospheric boundary layer turbulence needs to account for different scales of motion since turbulence occurs over a wide range of scales from dissipation scales to very large scale motion on the order of tens of kilometers. Using sonic anemometer data from the 60m tower from the CASES99 field experiment near Leon, KS, the variances and covariances are expressed as sums of the variances and covariances due to motions at a range of temporal scales through the use of a multiresolution decomposition. The temporal scales are converted to spatial scales by multiplying by the mean wind value. Turbulent kinetic energy (TKE) has the most energy in scales of motion about 600m at a location 50m agl. This peak is broad; the width at half max covers a range of turbulence scales from 20m to 2500m (1.5 sec - 3.5 min). Individual variances show peak energies at different scales; the vertical variance having peak energy at smaller scales than the TKE peak scales, and streamwise variances having peak energy at larger scales. Analysis of all three covariances shows that the assumption of 2D flow is not a good approximation for the 50m agl.

9:44AM G27.00009 Estimating Wind Turbine Inflow Using Sparse Wind Data[1] , RAJ RAI, JONATHAN NAUGHTON, University of Wyoming — An accurate spatially and temporally resolved estimation of the wind inflow under various atmospheric boundary layer stability conditions is useful for several applications relevant to wind turbines. Estimations of a wind inflow plane in a neutrally stable boundary layer using sparse data (temporally resolved but spatially sparse, and spatially resolved but temporally sparse) has shown good agreement with the original data provided by a Large Eddy Simulation. A complementary Proper Orthogonal Decomposition-Linear Stochastic Estimation (POD-LSE) approach has been used for the estimation in which the POD identifies the energetic modes of the flow that are then used in estimating the time dependent flow-field using LSE. The applicability of such an approach is considered by simulating the estimation of the wind inflow using data collected in the field. Modern remote measurement approaches, such as Lidar (Light detection and ranging), can sample the wind at the multiple locations, but cannot sufficiently resolve the inflow in space in time that is required for many wind turbine applications. Since inflow estimations using the POD-LSE approach can simultaneously provide spatial and temporal behavior, the use of the approach with field data for better understanding the characteristics of the wind inflow at a particular site under different atmospheric conditions is demonstrated.

[1]Support from a gift from BP is acknowledged.

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G28 Swimming Efficiency Ballroom II

8:00AM G28.00001 For free swimming the efficiency and economy optima are the same , HOSSEIN HAJ-HARIRI, University of Virginia — Most computational and experimental studies into the "optimality" of fins or wings are based on the placement of a model in a uniform stream and finding an optimum of either efficiency or economy. This approach is easy, but inherently inconsistent: any efficiency other than zero implies the presence of thrust, which is then incompatible with uniform speed. The proper way to reconcile the two is to assume the presence of sufficient parasitic drag to balance the thrust. But then different wings are implicitly attached to different bodies, and the optima are over a range of unrelated bodies. The consistent way to address optimization is in the context of free swimming. In this work a simple theoretical model based on a heaving and pitching plate is used to investigate the implications of free swimming. In particular, performance is optimized over the manifold of constant average thrust. Once constrained to this manifold, then the efficiency and economy optima are collocated. This simple model can predict the results of our prior computations for flexible wings. More importantly, the model details the interplay between the circulatory and non-circulatory lift/thrust, and can predict the motion of whale tails. The phase of pitch and heave work themselves out so as to keep the motion on the aforementioned manifold. These results have significance to swimming and to insect flapping in air where added mass has considerable effect.

8:13AM G28.00002 ABSTRACT WITHDRAWN –

8:26AM G28.00003 Hydrodynamic Wake Resonance as an Underlying Principle of Efficient Unsteady Propulsion[1] , KEITH MOORED, Princeton University, HOSSEIN HAJ-HARIRI, University of Virginia, PETER DEWEY, BIRGITT BOSCHITSCH, ALEXANDER SMITS, Princeton University — In this study, three dimensional wake structures are generated by an actively flexible robotic elliptical fin. Particle image velocimetry is used to characterize and quantify the wake and to extract time averaged velocity profiles. A linear spatial stability analysis is performed on the velocity profiles to find the frequency of maximum spatial growth, i.e. the resonant frequency of the time averaged jet. It is found that (1) optima in propulsive efficiency occur when the driving frequency of a flapping fin matches the resonant frequency of the jet profile, (2) there can be multiple wake resonant frequencies and modes corresponding to observed multiple peaks in efficiency, (3) observed wake structures are largely linear phenomena and (4) wake patterns are strongly influenced by the nearest resonant mode. Multiple peaks in efficiency and transitions in the wake structure can be explained by this theoretical framework. Surprisingly, the analysis, although one-dimensional, captures the performance exhibited by a three-dimensional propulsor, showing the robustness and broad applicability of the technique.

[1]Supported by ONR MURI

8:39AM G28.00004 Identifying optimal vortex spacing for swimming and flying animals[1] , PETER A. DEWEY, KEITH W. MOORED, DANIEL B. QUINN, ALEXANDER J. SMITS, Princeton University — Swimming and flying animals generate thrust by creating an unsteady vortex wake through the oscillation of their appendages. To determine the vortex spacing that maximizes propulsive efficiency, a finite core vortex array model was developed to compute the unsteady velocity field generated by vortex streets representative of bio- inspired propulsion. The model systematically varies the streamwise and transverse spacing between vortex cores to determine the time averaged velocity field induced by a reverse von Karman vortex street and a uniform freestream velocity. Experimental particle image velocimetry was conducted in the wake of a rigid pitching panel to determine the size and strength of the vortex cores to input to the model. Viscosity is accounted for by assuming a Gaussian vorticity distribution around the vortex core. A linear spatial stability analysis was performed on the computed velocity profiles to determine which vortex configuration leads to efficient propulsion. Here it is assumed that efficient propulsion proceeds when the driving frequency of the vortex street matches the resonant frequency of velocity jet.

[1]Supported by ONR MURI Grant N00014-08-1-0642.

8:52AM G28.00005 ABSTRACT WITHDRAWN –

9:05AM G28.00006 Propulsive performance of a flapping foil in a hydrodynamic tunnel: direct force measurements[1] , RAMIRO GODOY-DIANA, VERONICA RASPA[2], PMMH UMR7636 CNRS; ESPCI ParisTech; UPMC; U. Diderot Paris 7 — The study of simplified flapping wings has received much attention in the past two decades because of the renewed interest in biomimetic locomotion at intermediate Reynolds numbers ($10-10^4$). Recent works from our group have been devoted to the study of a pitching foil system in a hydrodynamic tunnel, exploiting particle image velocimetry first to characterize the transitions in the flow around the foil as a function of the flapping parameters (amplitude and frequency), and second to investigate the effect of flexibility. Here we report on our first results with an improved experimental setup where the pitching foil mechanism is mounted on a mechanical balance that allows us to have a time-resolved direct force measurement using an LVDT displacement sensor. We compare the performance of two different foils, one rigid and one flexible that have been previously characterized. We analyze the time-correlation of the thrust measurement with the instantaneous angular position of the foil, as well as the mean values of the force signals.

[1]We acknowledge support from the French National Research Agency through project No. ANR-08-BLAN-0099
[2]Postdoctoral fellow of the Research in Paris programme

9:18AM G28.00007 Optimal shapes for self-propelled swimmers , PETROS KOUMOUTSAKOS, WIM VAN REES, MATTIA GAZZOLA, ETH Zürich — We optimize swimming shapes of three-dimensional self-propelled swimmers by combining the CMA- Evolution Strategy with a remeshed vortex method. We analyze the robustness of optimal shapes and discuss the near wake vortex dynamics for optimal speed and efficiency at Re=550. We also report preliminary results of optimal shapes and arrangements for multiple coordinated swimmers.

9:31AM G28.00008 Effect of Vehicle Configuration on the Performance of a Submersible Pulsed-Jet Vehicle at Intermediate Reynolds Number[1] , P.S. KRUEGER, J.T. NICHOLS, SMU — Recent results have demonstrated that pulsed jet propulsion can achieve propulsive efficiency greater than that for steady jets when short, high frequency pulses are used, and the pulsed-jet advantage increases as Reynolds number decreases into the intermediate range ($\sim$50). An important aspect of propulsive performance, however, is the vehicle configuration. The nozzle configuration influences the jet speed and, in the case of pulsed-jets, the formation of vortex rings with each jet pulse, which have important effects on thrust. Likewise, the hull configuration influences the vehicle speed through its effect on drag. To investigate these effects, several flow inlet, nozzle, and hull tail configurations were tested on a submersible, self-propelled pulsed-jet vehicle ('Robosquid' for short). In terms of propulsive efficiency, changing between forward and aft-facing inlets had little effect, but changing from a smoothly tapered aft hull section to a blunt tail increased propulsive efficiency slightly due to reduced drag for the blunt tail configuration at intermediate Re. Sharp edged orifices also showed an advantage over smooth nozzles.

[1]This material is based on the work supported by the National Science Foundation under grant no 0347958.

9:44AM G28.00009 The Effect of Caudal Fin Shape on the Hydrodynamics of Swimming[1] , MOHSEN DAGHOOGHI, IMAN BORAZJANI, Mechanical and Aerospace Engineering Department, University at Buffalo SUNY — The caudal fin is thought to be the main thrust generator in body/caudal fin swimmers because the largest undulations occur at the caudal fin. The shape of the fin could possibly be one of the most important factors in thrust generation for such swimmers. However, investigating this experimentally is quite challenging due to the issues in controlling and measuring forces on different appendages of live fish. We can investigate the effect of caudal fin shape through controlled numerical simulations. We construct virtual swimmers with different caudal fin shapes but with the same projected area. We attach trapezoidal and heterocercal shapes of caudal fins (e.g. observed in trouts and sharks, respectively) to a mackerel body and test these swimmers beside the original mackerel with a hemocercal tail. We prescribe the same carangiform kinematics to all virtual swimmers and carry out self-propelled simulations under similar conditions, i.e., the undulations are prescribed while motion of the center of mass is calculated. The simulations are continued until the quasi-steady state is reached, in which the swimmers are compared in terms of different performance measures.

[1]This work was partly supported by the Center for Computational Research at the University at Buffalo.

9:57AM G28.00010 Effects of shape on the thrust performance and vortical structure of flapping plates[1] , XI-YUN LU, GAO-JIN LI, University of Science and Technology of China — Three-dimensional unsteady hydrodynamics of flapping plates with different shapes are numerically investigated using the lattice Boltzmann and immersed boundary method. The plate shape is quantitatively described by the area moments. The thrust exhibits a monotonous increase with the area moments. Two typical regimes are identified. One is un-linear relation between the thrust and the area moment for lower area moment and the other is linear relation for larger area moment. Moreover, the vortical structures in the wake are usually composed of two rows of vortex rings formed by the vortices shed from the leading and trailing edges as well as the two tip sides. As the strength of the vortices changes with the plate shape, the topology of the vortical structures is closely related to the plate shape. Further, the intrinsic relationship between the hydrodynamics and the vortical structures in the wake is analyzed using some unconventional force expressions and simplified vortex ring models.

[1]This work was supported by NSFC (No. 10832010).

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G29 Biofluids: Cellular III: Vesicles Ballroom III

8:00AM G29.00001 Molecular Transport across Small Blood Vessels[1] , AXEL GUENTHER, ZHAMAK ABDIDEZFOOLI, SASCHA PINTO, STEFFEN-SEBASTIAN BOLZ, University of Toronto — Small blood vessels dominate molecular transport of small and large molecules due to their large cumulative surface area. However, systematically probing specific transport mechanisms under well defined and physiologically meaningful conditions is challenging. We present the first microfluidic approach that allows the investigation of molecular transport across intact small blood vessels. Functionally intact mouse mesenteric arteries (150-300 micrometers in diameter) are perfused with fluorescent markers. The permeation rate across the vessel wall is locally deduced by laser scanning confocal microscopy and particle image velocity measurements in combination with a transport model, and by a global fluorescence spectroscopy measurement.

[1]NSERC I2I, CFI, ORF, NSERC CREATE MATCH

8:13AM G29.00002 The Brownian dynamics of a lipid vesicle , HONG ZHAO, ERIC SHAQFEH, Stanford University — We propose a Brownian dynamics simulation method to study the motion of a lipid vesicle in a suspending fluid. The lipid bilayer of the vesicle is modeled as a two-dimensional fluidic membrane with bending resistance and a high local area dilatational modulus. Using a Stokes flow boundary integral equation method, we calculate the grand mobility tensor of the deformable membrane at arbitrary viscosity ratio between the fluid inside and outside the vesicle. The fluctuation–dissipation theorem, as well as the drifting due to the configuration dependence of the diffusivity tensor, are rigorously accounted for. The flow transitions (tank-treading, tumbling and trembling) and the particle stresslet for an athermal vesicle in a simple shear flow calculated by the present method are in good agreement with existing results by a spectral boundary integral equation method. We demonstrate that the effect of Brownian motion is most significant for a tumbling vesicle. The thermal fluctuation tilts the vesicle off the shear plane, and the disruption of the originally two-dimensional tumbling orbit results in a qualitatively different three-dimensional "wobbling" motion. The change to the suspension rheology due to the altered dynamics is discussed.

8:26AM G29.00003 Dynamics of Lipid Bilayer Vesicles and Droplets in DC Electric Fields , LANE MCCONNELL, Northwestern University, PETIA VLAHOVSKA, Brown University, MICHAEL MIKSIS, Northwestern University — Closed lipid bilayers (vesicles) serve as a model system to study the mechanics of the biomembranes encapsulating cells and cellular organelles. We present a numerical investigation using the Boundary Integral Method of the dynamics and stability of a charge-free lipid bilayer vesicle in a uniform DC electric field. The lipid membrane is modeled as a zero-thickness, capacitive, area-incompressible interface, and bulk fluids are assumed to be leaky dielectrics. Vesicle shape is determined by balancing the electric, hydrodynamic, bending, and surface tension stresses along the interface. Investigations of vesicle response to electric fields are limited, but recent small deformation analysis has revealed several interesting phenomena not observed with droplets, including transitions from oblate to prolate shapes. Our numerical investigations highlight the differences in the behavior of vesicles and drops due to the capacitive nature of the bilayer membrane.

8:39AM G29.00004 Vesicle Shape Transformations Driven by Active and Spontaneous Lipid Flip-flop , THOMAS POWERS, Brown University, ELNAZ BAUM-SNOW, University of Connecticut Health Center — The lipid composition of cell membranes is created and maintained in part by flippases, enzymes that translocate lipid molecules from one layer of the bilayer membrane to the other. We study how lipid translocation can affect membrane shape, using a cylindrical vesicle as a simple model system. For a short pulse of flippase activity, in which a fraction of lipids are flipped from one layer to the other, we calculate the fraction of flipped lipids that makes the cylinder unstable to a periodic modulation in its radius, as well as the growth rate of perturbations of different wavenumber. We also study the cases of continuous flippase activity and spontaneous flip-flop.

8:52AM G29.00005 Dynamics and phase diagrams for highly non-spherical vesicles in shear flow computed with Loop subdivision surfaces[1] , ANDREW SPANN, HONG ZHAO, ERIC SHAQFEH, Stanford University — Vesicles, particularly those with a low volume to surface area ratio, are challenging to simulate due to the presence of a surface incompressibility constraint and a bending energy that requires a highly accurate estimate of curvature. A boundary integral method based on Loop subdivision surfaces on an unstructured mesh is used to compute phase diagrams and stress dynamics for highly non-spherical vesicles in shear flow. In addition to the most commonly studied prolate family of vesicles, we also investigate the biconcave and stomatocyte shapes. We chronicle the decrease in the viscosity ratio threshold needed to trigger transition between the regimes of tank treading, trembling, tumbling, and kayaking as the reduced volume ratio of a prolate vesicle decreases. For biconcave shapes, we observe three regimes: conversion to prolate, tank treading, and tumbling. We find that biconcave tumbling near the critical viscosity ratio is not merely a rotation motion and includes noteworthy stretching in the vorticity direction.

[1]Funded by NSF GRFP and Stanford Graduate Fellowship

9:05AM G29.00006 Electrohydrodynamics of bilayer membranes[1] , PAUL SALIPANTE, Brown University, RUMIANA DIMOVA, Max-Planck Institute of Colloids and Interfaces, PETIA VLAHOVSKA, Brown University — Membranes that encapsulate cells and internal cellular organelles are composed primarily of lipid bilayers. Biomimetic membranes assembled from polymers are used as vectors for targeted drug delivery. We investigate the deformation and stability of fluid membranes made of lipids or polymers in uniform electric fields. A frequency dependent shape deformation of vesicles (closed membranes) in AC fields elucidates the capacitive nature of the membrane and provides a new experimental method for measuring membrane capacitance. Compared to lipid membranes, we find that polymer membranes have an order of magnitude lower capacitance, which correlates with their larger thickness. Upon application of the electric field, the dynamic response of the vesicle is sensitive to membrane viscosity, while the steady state shape is governed by membrane tension and bending stiffness. Strong DC pulses, typically used in cell electroporation, is shown to induce an instability in both lipid and polymer membranes. The instability leads to vesicle collapse, where the timescale of collapse shows a $t \sim 1/E^2$ dependence.

[1]Supported by NSF grants CBET-1117099 and OISE-1032280.

9:18AM G29.00007 The relaxation of tweezed giant unilamellar vesicles , WOLFGANG LOSERT, Department of Physics, Institute for Physical Science and Technology, University of Maryland, College Park, USA, HERNAN ZHOU, Physics Department, Eindhoven University of Technology, BEATRIZ BURROLA GABILONDO, Department of Physics, Institute for Physical Science and Technology, University of Maryland, College Park, USA, WILLEM VAN DE WATER, Physics Department, Eindhoven University of Technology — We study the shape relaxation of spherical giant unilamellar vesicles which have been deformed far from equilibrium into ellipsoids using optical tweezers. The relaxation back to a sphere is determined by elastic constants of the vesicles, and their excess area, as well as by low Reynolds number fluid flow. The properties of each vesicle are learned from observing its shape fluctuations in thermal equilibrium. We will critically evaluate the spectroscopic techniques used to find those properties. The relaxation time of the stretched vesicles could be compared favorably to a simple formula which encompasses the joint effect of membrane rigidity and fluid flow. The time constant of the stretched vesicle is slower than that of its thermal fluctuations, which agrees with a recent theory, however, it is still one order of magnitude faster than predicted.

9:31AM G29.00008 Viscoelastic properties of vascular endothelial cells exposed to uniaxial stretch[1] , KATHRYN OSTERDAY[2], MAE Dept, THOMAS CHEW, PHILLIP LOURY, JASON HAGA, Dept of Bioengineering, JUAN C. DEL ALAMO, MAE Dept., SHU CHIEN, Depts. of Bioengineering and Medicine, University of California San Diego — Vascular endothelial cells (VECs) line the interior of blood vessels and regulate a variety of functions in the cardiovascular system. It is widely accepted that VECs will remodel themselves in response to mechanical stimuli, but few studies have analyzed the mechanical properties of these cells under stretch. We hypothesize that uniaxial stretch will cause an anisotropic realignment of actin filaments, and a change in the viscoelastic properties of the cell. To test this hypothesis, VECs were grown on a thin, transparent membrane mounted on a microscope. The membrane was stretched, consequently stretching the cells. Time-lapse sequences of the cells were taken every hour with a time resolution of 10 Hz. The random trajectories of intracellular endogenous particles were tracked using in-house algorithms. These trajectories were analyzed using a novel particle tracking microrheology formulation that takes into account the anisotropy of the cytoplasm of VECs.

[1]Supported by NSF CBET-1055697 CAREER Award (JCA) and NIH grants BRP HL064382 (SC), 1R01 HL080518 (SC).
[2]Supported by NSF-GRFP

9:44AM G29.00009 The evolution of a sessile vesicle's shape during desiccation processes[1] , MAURICE BLOUNT, MICHAEL MIKSIS, STEPHEN DAVIS, Northwestern University — We present a model that describes the shape evolution of a vesicle that is attached to a substrate. During desiccation processes the vesicle contains an aqueous solution of sugar and is surrounded by an aqueous solution of sugar of higher concentration. Transport of water across the vesicle's semi-permeable membrane is driven by concentration and pressure gradients applied across it. As water is drawn out of the vesicle, its volume decreases but its surface area is conserved owing to the membrane's incompressibility. The consequent buckling of the membrane is impeded by its bending stiffness and by the viscous stresses in the flow that are generated by the membrane's motion. We use our model to describe the various physical nonlinear effects that evolve during the process.

[1]Supported by NIH Grant 5R01GM086886.

9:57AM G29.00010 Numerical Analysis of Vesicle Dynamics in Linear Shear Flow[1] , ALIREZA YAZDANI, PROSENJIT BAGCHI, Rutgers University — There has been an ongoing debate in the vesicle research community with regard to the dependence of the vesicle dynamics on the controlling parameters, namely, the shear rate, viscosity ratio, and vesicle excess area. Theoretical works by Misbah and coworkers [e.g., Kaoui et al, Phys Rev E, **80**, 061905 (2009)] predict that the dynamics depends on all three parameters, whereas experimental works by Steinberg's group [e.g., Zabusky et al, Phys Fluids, **23**, 041905 (2011)] suggest only two parameters. In order to provide further insight, we consider 3D numerical simulations using a front-tracking method. We model the membrane bending resistance using the Helfrich formulation [Zhong-can & Helfrich, Phys Rev A, **39**, 5280 (1989)], and the area dilatation using the strain energy function developed by Skalak et al [Biophys J. **13**, 245, (1973)]. The simulations successfully predict three different dynamical regimes, namely, the tank-treading, vascillating-breathing, and tumbling. Quantitative comparisons are made with the theoretical results as well as the experimental data. We find that the tank-treading inclination angle and the tank-treading-to-tumbling transition threshold is weakly dependent on the shear rate. We further provide phase diagrams, and discuss the role of the controlling parameters on the transition.

[1]Supported by NSF

Monday, November 21, 2011 8:00AM - 10:10AM –
Session G30 Biofluids: Arterial Flows: Curving, Branching, and Stents Ballroom IV

8:00AM G30.00001 Structures in secondary flow under simple harmonic inflow in a 180 degree curved pipe model of an artery[1] , AUTUMN L. GLENN, SARAH L. SEAGRAVE, KARTIK V. BULUSU, MICHAEL W. PLESNIAK, The George Washington University — Inward centrifuging of fluid in a 180 degree curved pipe leads to development of secondary flow vortical structures. These Dean's vortices have been widely studied in steady flows. Complex secondary flow structures were observed under (unsteady) physiological flow forcing associated with the cardiac cycle, as well as simple harmonic forcing. These structures were investigated under several simple harmonic inflow conditions with phase-locked 2-D PIV measurements to examine the formation of coherent structures in the secondary flow. Experimental velocity field data were acquired at various cross-sectional planes along the bend. Multiple vortex pairs were observed at 90 degrees into the bend for all waveforms investigated. The overarching goal of this study is to understand the effect of driving waveform characteristics, i.e. period, flow acceleration, etc. on secondary flow morphologies and to characterize these morphologies in terms of dimensionless parameters describing the flow.

[1]Supported by the National Science Foundation under Grant No. CBET-0828903

8:13AM G30.00002 A numerical study of steady flow through a curved tube with wavy walls ,

CHEKEMA PRINCE, MINGYAO GU, SEAN PETERSON, University of Waterloo — Flow through curved tubes has been studied for nearly a century owing to the practical industrial applications and general academic interest. More recently, interest in curved tubes has resurfaced due to the ubiquity of curvature in the vasculature and the resulting need to accurately model arterial vessels. Previous studies have focused primarily on circular cross sections and the roles of the Dean number and curvature ratio on the flow physics. In this study we examine the effect of wavy walls, that is, axially aligned ribs extending the length of the tube, on steady flow through mildly and finite curved tubes using computational fluid dynamics. Analytical work on the subject has been limited to low Dean numbers and small bump heights, thus we primarily focus on the impact of higher Dean number with large protrusions on the flow physics. The results are compared with those in circular cross section tubes at the same Dean number. Particular attention is paid to flow characteristics of interest in the vasculature, such as wall shear stress, that have been shown to stimulate biochemical pathways that trigger cell growth.

8:26AM G30.00003 Secondary flow structure detection using boundary tracing under physiological inflow through a bent pipe model for curved arteries[1] ,

SHADMAN HUSSAIN, KARTIK V. BULUSU, AUTUMN L. GLENN, The George Washington University, FANGJUN SHU, New Mexico State University, MICHAEL W. PLESNIAK, The George Washington University — Experimental data from an investigation of secondary flows in a bent artery model under physiological inflow conditions were obtained using the PIV technique. Continuous wavelet transforms were used to resolve coherent vortical structures in cases with and without an idealized stent model. Boundaries of these structures were traced using MATLAB functions to estimate their circulation and scale. Vortical structures of different scales, possessing approximately equal strength were observed in histograms of phase-locked measurements with the stent model. A circulation threshold was established to facilitate identification of smaller-scale, high-circulation secondary flow vortical structures. A parametric study of circulation thresholding was performed to gain insight into the scales associated with secondary flow structures in curved artery models.

[1]Supported by the National Science Foundation under Grant No. CBET-0828903.

8:39AM G30.00004 Continuous wavelet analysis of stent-induced perturbations in a bent pipe model for curved arteries[1] ,

KARTIK V. BULUSU, AUTUMN L. GLENN, The George Washington University, FANGJUN SHU, New Mexico State University, SHADMAN HUSSAIN, MICHAEL W. PLESNIAK, The George Washington University — Secondary flow vortical structures were observed in a 180 degree circular bend under physiological flow conditions with a stent model installed upstream of the bend. Phase-locked 2-D PIV measurements were made at various cross-sectional planes along the bend. Stent-induced perturbations led to a transient flow regime with a multiplicity of vortical patterns initiated during the deceleration phase of the systolic peak (starting at t/T=0.21). An exploratory investigation of vortical scale-count metrics from continuous wavelet transforms, was performed using a Ricker wavelet. The metrics highlight the evolution of a pair of ordered, coherent, high-circulation, counter-rotating vortical structures (at t/T=0.21) into multiple, disordered, low-circulation, coherent vortical structures (by t/T=0.30). The overarching goal of this study is to create a regime map of secondary flow morphologies based on the driving physiological waveform. An approach to develop a regime map using vortical scale-count metrics is outlined.

[1]Supported by the National Science Foundation under Grant No. CBET-0828903

8:52AM G30.00005 The effect of waveform shape on the flow in endovascular stents ,

AMIRREZA ROUHI, MATTHEW FORD, UGO PIOMELLI, Queen's University, PAVLOS VLACHOS, VPISU — We studied the effects of the imposed waveform on the flow in an idealized stented artery. Two stents were used in this study, both based on commercially available geometries. The Navier-Stokes equations were solved using a cartesian staggered code, and the stents were modelled via an immersed-boundary method. The Reynolds number and Womersley number are 160 and 2.09 respectively. The presence of flow reversal results in the formation of vortices between the stent struts, which detach from the wall and migrate toward the center of the channel. Truncated series representations of the waveform shape were tested to determine the effect of the waveform detail on the flow dynamics. A reduction in the number of Fourier modes did not affect the time-averaged wall shear stress or the distribution of the Oscillatory Shear Index. The formation of vortical structures and their kinematics were insensitive to the geometrical detail. We examined cases with decreased percentages of flow reversal, achieved by increasing the mean flow rate while maintaing the waveform shape. We initially observed two vortex formations and liftoffs per cycle. As the duration of flow reversal was decreased, the vortex creation and migration occurred once with the vortex remaining suspended above the stent wires before disappearing. When the flow was unidirectional, no vortex creation was observed.

9:05AM G30.00006 Hemodynamics and transient flow reversal in real deployed stents ,

RALPH METCALFE, MIRCEA IONESCU, University of Houston — Restenosis rates caused by neointimal hyperplasia are relatively high ($\sim 30\%$) after stent implantation in stenosed arteries. The flow around stent struts under steady and unsteady conditions using computational hemodynamics (CHD) was studied to identify contributing factors to the formation of low and oscillating wall shear stress regions that have been shown to promote endothelial dysfunction and atherosclerotic plaque formation in arteries. Datasets of the Neuroform, BxVelocity, and Taxus stents deployed in straight polymer tubes were obtained from high resolution micro computed tomography. Finite volume CHD simulations of steady and unsteady flow with and without flow reversal were performed. Stagnation zones were noticed adjacent to the strut junctions as the flow enters and exits the stent cells. The stagnation zones were larger in the case of the stents with larger strut diameter (BxVelocity, Taxus), wider strut junctions and larger angles between the struts. Unsteady flow simulations showed enhanced flow reversal with thicker struts and large regions of recirculation flow developing inside the stent at Reynolds numbers higher than 200. It was shown that alterations in blood flow due to real stent deployment (strut prolapse, junction misalignment) cannot be captured with computer generated stent models, that stent specific geometry, and time dependent flow effects can locally alter the wall shear stress and stagnation zones.

9:18AM G30.00007 ABSTRACT WITHDRAWN —

9:31AM G30.00008 Effect of rheological property on blood flow in vertebral artery branch ,

TAEGEE MIN, S&H Co. Ltd., MYUNGJOON KIM, TAESUNG KIM, Sung Kyun Kwan University, O-KI KWON, Seoul National University — Blocking of an artery is one of mechanisms for cerebral stroke development. If an important cerebral artery is occluded by any reason and if there is no sufficient collaterals, tissue ischemia occurs at brain tissues distal to the occluded artery, which is a well known clinical situation. However, in practice, ischemia or hypoperfusion has also been observed through the branches proximal to the occluded artery. The unexpected "proximal ischemia" is not yet known, from which patients could suffer serious complications. In the present study, two patient cases are presented to elucidate this phenomenon from the view point of fluid dynamics, especially with emphasis on the role of rheology in hemodynamics.

9:44AM G30.00009 ABSTRACT WITHDRAWN —

9:57AM G30.00010 Is the human left ventricle partially a fractal pump? , BRANDON MOORE, LAKSHMI DASI, Colorado State University, Fort Collins CO — Ventricular systolic and diastolic dysfunctions represent a large portion of healthcare problems in the United States. Many of these problems are caused and/or characterized by their altered fluid-structure mechanics. The structure of the left ventricle in particular is complex with time dependent multi-scale geometric complexity. At relatively small scales, one facet that is still not well understood is the role of trabeculae in the pumping function of the left ventricle. We utilize fractal geometry tools to help characterize the complexity of the inner surface of the left ventricle at different times during the cardiac cycle. A high-resolution three dimensional model of the time dependent ventricular geometry was constructed from computed tomography (CT) images in a human. The scale dependent fractal dimension of the ventricle was determined using the box-counting algorithm over the cardiac cycle. It is shown that the trabeculae may indeed play an integral role in the biomechanics of pumping by regulating the mechanical leverage available to the cardiac muscle fibers.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H1 Geophysical Flows: Oceanography III 301

10:30AM H1.00001 The Influence of Crosswind Tidal Currents on Langmuir Circulation in a Shallow Ocean , TOBIAS KUKULKA, University of Delaware, ALBERT PLUEDDEMANN, JOHN TROWBRIDGE, Woods Hole Oceanographic Institution, PETER SULLIVAN, National Center for Atmospheric Research — Langmuir circulation (LC) is a turbulent process driven by wind and surface waves that plays a key role in transferring momentum, heat, and mass in the oceanic surface layer. On the coastal shelves the largest scale LC span the whole water column and thus couple the surface and bottom boundary layers and enhance turbulent mixing. Observations and large eddy simulations (LES) of a shallow coastal ocean demonstrate that these relatively large scale Langmuir cells are strongly influenced by crosswind tidal currents. Two mechanisms by which crosswind tidal shear may distort and disrupt Langmuir cells are proposed. The first mechanism involves cell shearing due to differential advection across the whole cell. For the second mechanism, mid-depth vertical LC currents advect sheared mean crosswind current, leading to the attraction of up- and downwelling regions, so that LC cells are unsustainable when both regions overlap. Scaling arguments indicate that LC cells are more susceptible to crosswind shear distortion for smaller LC surface velocity convergence and greater cell aspect ratio (vertical to horizontal LC scale), which is consistent with the results obtained from the observations and LES.

10:43AM H1.00002 Multiscale asymptotic analysis and numerical simulation of the wind-driven ocean surface boundary layer[1] , ZIEMOWIT MALECHA, GREG CHINI, University of New Hampshire, KEITH JULIEN, University of Colorado, Boulder — A primary challenge in physical oceanography is to understand the interaction between small-scale vertical mixing processes in the upper ocean, such as Langmuir circulation (LC), and submesoscale eddies, fronts, and their associated instabilities. This problem is computationally challenging because LC is strongly non-hydrostatic, only weakly affected by the Earth's rotation and density stratification, and has length scales commensurate with the ocean surface boundary layer, O(50) m. In contrast, submesoscale flows are approximately hydrostatic, strongly affected by Coriolis accelerations and density stratification, and have O(10) km lateral scales. In this investigation, we take a first step toward developing a physically consistent and computationally efficient model of this inter-scale coupling using multiscale asymptotic analysis and multiscale pseudospectral numerical simulations. We have achieved over an order of magnitude acceleration of our computations relative to brute-force simulations using a single-scale algorithm.

[1]GC gratefully acknowledges support for this work through NSF CMG grant 0934827.

10:56AM H1.00003 Beyond mean-field theory: An asymptotically-inspired single-mode model of Langmuir circulation , ZHEXUAN ZHANG, GREG CHINI, University of New Hampshire, KEITH JULIEN, University of Colorado, Boulder — Mean-field theory for convecting flows dates back to the pioneering work of Spiegel (1962), Herring (1963, 1964, 1966), and others. In the classical approach, all fields are decomposed into horizontally-averaged and fluctuating components. The approximation involves retaining only those fluctuating-fluctuating nonlinearities in the governing equations that modify the mean fields. In laterally periodic geometries, the resulting mean-field PDEs admit single horizontal (Fourier) mode solutions, reducing the problem to a set of PDEs in the vertical (boundary-normal) coordinate and time. In this investigation, we develop a single horizontal-mode reduction of the Craik–Leibovich (CL) equations governing Langmuir circulation (LC). Unlike classical mean-field theory, however, we retain important but spatially localized fluctuating-fluctuating nonlinearities in our reduced model. The spatial structure of the basis functions and the choice of modes on which to project the dynamics are guided by the matched asymptotic analysis of fully nonlinear LC by Chini (2008). As in mean-field theory, our single-mode reduction yields a set of 1D PDEs, which are readily solved numerically. We demonstrate the efficacy of our reduced model via quantitative comparisons with full pseudospectral numerical simulations of the downwind-invariant CL equations.

11:09AM H1.00004 Influence of a crossed tidal current on a wind shear driven flow in shallow water with and without wave forcing by means of LES.[1] , GUILLAUME MARTINAT, Old Dominion University, ANDRES TEJADA-MARTINEZ, University South Florida, CHESTER E. GROSCH, Old Dominion University — Turbulent generated by tidal current and wind shear is important because it controls heat and mass transfer at the air-sea interface. Understanding the turbulence dynamics of this class of flows presents complications because of the presence of a free surface, the necessity of including the flow interaction with a solid bottom.The intent of the LES discussed here is to simulate the turbulent flow on shallow shelves having turbulence scales of O(100) m in the horizontal and depths in the range of 10 to 50 m. On these scales the turbulent flow is homogenous in the horizontal. The purpose of our calculations is to elucidate the structure of the turbulence dynamics as driven by pressure gradient (tidal) flow crossed with a surface stress driven flow both imposing an equal stress on the solid bottom, with and without wave forcing.

[1]Support from NSF award: 0927724 and Teragrid project: OCE100008 is aknowledged.

11:22AM H1.00005 LES of scalar transport in wave and wind-driven flows with large-scale structures , CIGDEM AKAN, ANDRES E. TEJADA-MARTINEZ, University of South Florida, CHESTER E. GROSCH, Old Dominion University — Surface scalar (mass) transport results from large-eddy simulation (LES) of wind-driven flow with Langmuir circulation are presented. Wave-current interaction gives rise to Langmuir turbulence characterized by Langmuir circulation (LC) consisting of a spectrum of scales of counter rotating vortices roughly aligned in the direction of the wind. The typical crosswind length scale of the smallest observed vortices is on the order of several centimeters when the wind begins to blow over a quiescent interface and short capillary waves first appear. The crosswind length scale of the largest vortices reaches up to tens of meters under sustained winds and longer waves. Two different types of flows will be analyzed. The first type is characterized by small scale (centimeter scale) LC and the second type is characterized by large scale LC in shallow water where the cells span the entire water column. Both types of flows exhibit increases in surface mass transfer velocity (efficiency) due to the presence of LC. Statistical analysis of LES variables will be presented highlighting the differences between small-scale and large-scale LC structures and their impact on near-surface scalar transport.

11:35AM H1.00006 Lateral mixing of passive scalars around porous obstacles in tidal flows ,

HYEYUN KU[1], SUBHAS VENAYAGAMOORTHY, Colorado State University — In this study, we use high-resolution two-dimensional numerical simulations to study mixing and transport of passive scalars around porous cylindrical obstacles in tidal flows. A submerged obstacle blocks the flow (partially or fully depending on its porosity) causing deceleration of the flow, shedding of vortices and the formation of a downstream wake. The lateral distribution of the passive scalar is quantitatively examined by calculating the lateral mixing coefficient as a function of three non-dimensional parameters namely; the non-dimensional drag coefficient, C_d (imparted by the obstacle); the ratio of the tidal to mean flow velocity amplitudes, $\eta = U_T/U_M$; and an tidal excursion parameter, $K = 2U_T/\omega D$, where ω is the forcing frequency and D is the diameter of an obstacle. The simulation results show that the lateral mixing is substantially enhanced due the combined effect of the drag exerted by the obstacle and the oscillatory flow compared to the classical dispersion in a uni-directional flow without drag. The results also highlight the complex dispersion patterns around submerged obstacles in oscillatory flows.

[1]PhD Candidate

11:48AM H1.00007 Numerical simulations of internal tides at a model ridge , NARSIMHA RAPAKA, BISHAKHDATTA GAYEN, SUTANU SARKAR, University of California San Diego — Direct and large eddy simulations are performed to study tidal flow over a model ridge. The Navier-Stokes equations are solved on a collocated grid using a mixed spectral-finite difference algorithm with a RK3-ADI time integration scheme. The effect of criticality parameter on the tidal conversion is studied in the laminar flow regime and compared with existing inviscid linear theory. Nonlinear processes become important at critical slope of the topography even for low excursion numbers. The effect of turbulence on tidal conversion is assessed under near-critical flow conditions by increasing the barotropic forcing amplitude. Phase dependence of turbulence statistics over a tidal cycle is analyzed at different locations on the ridge. Turbulence near the topography results in up to 25% reduction in the radiated tidal energy.

12:01PM H1.00008 ABSTRACT WITHDRAWN —

12:14PM H1.00009 Turbulent dynamics over evolving three-dimensional sandy rippled beds ,

ALLISON PENKO, JOSEPH CALANTONI, Naval Research Laboratory — Ripples on the seafloor affect the development of the wave bottom boundary layer impacting wave energy dissipation and sediment transport. Using a three-dimensional numerical model, SedMix3D, we simulate the complex turbulent flow of a fluid-sediment mixture over a rippled bed and the subsequent three-dimensional bed evolution. In general, results from SedMix3D are in excellent agreement with laboratory observations of time-dependent vorticity and swirling strength fields, ripple geometry characteristics, and statistically averaged flow quantities. We present simulations of evolving rippled beds of varying initial topographies with domains covering areas of the seafloor up to 12 cm x 24 cm with a vertical extent up to 24 cm. A significant difference between the spatial distribution of turbulence over three-dimensional ripple structures and two-dimensional parallel ripples is observed in the simulations. We also examine the effect of the turbulence on the maintaining and evolution of the two- and three-dimensional ripple structures.

12:27PM H1.00010 Transport and Mixing of the Oglio River Inflow into Lake Iseo , CHARLIE HOGG,

HERBERT HUPPERT, Institute of Theoretical Geophysics, University of Cambridge, UK, JORG IMBERGER, Centre for Water Research, University of Western Australia, Australia — The fate of river water entering a lake remains an uncertainty in many important limnological questions. These questions include how to improve standard water management practices and how lake ecosystems will change in future climate scenarios. This paper describes a field campaign carried out to understand the transport and mixing of a river inlet into Lake Iseo, a subalpine lake in Italy. We observed the low Froude number inflow to fall laterally after entering the lake. We suggest that this is caused by baroclinic acceleration. This laterally falling regime has not, to our knowledge, previously been described in the literature. In addition, measurements of a range of tracers were taken to find the dilution of the river after it had started to intrude into the lake. The tracers used were temperature, salinity, dissolved oxygen, coloured dissolved organic matter and turbidity. Our results found self-consistent mixing rates from the available natural tracers. These findings contribute added evidence and improve the understanding of what mechanisms cause mixing of river inflows.

Monday, November 21, 2011 10:30AM - 12:40PM —

Session H2 Compressible Flows I 302

10:30AM H2.00001 Bulk Viscosities of Low-Pressure Gases[1] , MARK CRAMER, Virginia Polytechnic Institute and State University — We provide estimates of the bulk viscosity of ideal gases using Tisza's formula and published values of the rotational and vibrational relaxation times. It is shown that, unlike the shear viscosities, the bulk viscosities take on a wide variety of numerical values and temperature variations. Common fluids having bulk viscosities which are hundreds to thousands of times larger than their shear viscosities are identified. We argue that the temperature variation of the bulk viscosity of many fluids will have a local maximum.

[1]This research was supported by NSF grant CBET-0625015.

10:43AM H2.00002 The Inviscid and Boundary Layer Approximations in Fluids with Large Bulk Viscosities[1] , FATEMEH BAHMANI, MARK CRAMER, Virginia Polytechnic Institute and State University — We re-examine the structure of classical inviscid steady flow over a finite body in Navier-Stokes fluids having bulk viscosities which are large compared to their shear viscosities. When the ratio of bulk to shear viscosity is of the order of the square root of the Reynolds number both the inviscid flow and boundary layer require corrections on the order of the inverse square root of the Reynolds number.

[1]This research was funded by NSF grant CBET-0625015.

10:56AM H2.00003 Detailed Simulations of Incident Shock Wave Development and Boundary Layer Growth in Shock Tube Facilities , YONG SUN, MATTHIAS IHME, University of Michigan, RALF DEITERDING, Oak Ridge National Laboratory Oak Ridge — The boundary layer growth after the incident shock in shock-tube systems results in the attenuation of the incident shock, which leads to uncertainties in measurements. Apart from few computational investigations, the current understanding about the boundary layer dynamics in shock tubes is largely derived from asymptotic analysis. To address this, detailed numerical simulations in realistic shock-tube systems at relevant operating conditions were conducted to quantify direct contributions of the boundary layer development on the incident shock wave and resulting deviations from ideal shock tube conditions. Both 3D results in cylindrical coordinates and 2D axis-symmetry results were compared with theoretical formulations for boundary layer developments. From this study, detailed characterization of boundary layer growth in realistic shock tube systems was obtained to validate potential models.

11:09AM H2.00004 Simulations of the Formation and Hydrodynamic Penetration of Micro-Shaped Charge Jets[1]

D. SCOTT STEWART, Mechanical Science and Engineering, University of Illinois, Urbana, IL 61801, FADY M. NAJJAR, Weapons & Complex Integration Directorate, Lawrence Livermore National Laboratory, MATTHEW SZUCK, NICK GLUMAC, Mechanical Science and Engineering, University of Illinois, Urbana, IL 61801 — An explosively formed shape charge jet can be generated by the action of a detonation in explosive charge that surrounds a hollow cone of metal, embedded in the explosive, that collapses the cone on the central axis in order to form a forward-going jet of metal. We discuss the results of a series of multi-material simulations for very small charges and cones, for which the thickness of the metal (copper) cone is less than 1/100th of an inch. We look at the ability of these micro-shaped charge jets ability to penetrate aluminum target blocks and compare against experiment. We examine the effects of shape defects in the cone liner and how they affect the penetration depth. The LLNL multi-physics hydrodynamic code ALE3D is used to carry out the simulations.

[1]DSS, MS and NG were supported by AFRL/RW AF FA8651-10-1-0004. FMN's work performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344. LLNL-ABS-491793.

11:22AM H2.00005 Underwater shock focusing by composite structures[1]

CHUANXI WANG, University of Southern Califronia, VERONICA ELIASSON, University of Southern California — Underwater explosions are threats to the structural integrity of naval vessels. In particular, if a convergent section is present on the vessel, the shock wave can focus and produce extremely high pressures near the focal region. Based on previous research on converging shock waves, a logarithmic spiral duct is considered to be an efficient shape to focus shock waves onto the focal region. Here, underwater shock tests on logarithmic spiral-shaped structures made of plastic, metal and fiber composites are conducted. High-speed schlieren photography is used to visualize the shock waves. Simultaneously, ultrafast pressure readings are recorded by laboratory- made pressure sensors, which are able to measure pressures up to 10 GPa. Comparisons between the various types of surrounding materials will be presented. The results can explore the use of composite materials in future marine applications.

[1]Supported by ONR through a MURI grant number N00014-06-1-0730.

11:35AM H2.00006 Large-Eddy Simulation of an over-expanded planar nozzle

BRITTON OLSON, SANJIVA LELE, Stanford University — Large-eddy simulation (LES) of an over-expanded planar nozzle is performed to elucidate the complex interaction between the turbulent boundary layer, the internal shock wave and the separated shear layer downstream of the shock. This numerical simulation seeks to model one the experiments of Papamoschou et.al. and shed light on the underlying physics of the shock-boundary layer interaction and the Free-Shock Separation (FSS) which is present in over-expanded supersonic nozzles. Simulation results compare well with those obtained from the experiment despite the modeling approximations. The LES captures the unsteady fluctuation of the shock wave as it interacts with the incoming turbulent boundary layer and the separated shear layer downstream. The richness of the simulation data allow for a more in-depth exploration of the underlying physics and have informed development of a reduced order model for approximating the large-scale dynamics of the complex system.

11:48AM H2.00007 Creating sharp features by colliding shocks on ion bombarded surfaces

MIRANDA HOLMES-CERFON, MICHAEL BRENNER, Harvard University — When a surface is bombarded with ions, it erodes at different rates depending on the slope of the surface, and intricate patterns can evolve. On large scales the surface slope evolves according to a conservation law, which can form shocks, while the type of shocks which can occur are governed by the small-scale fourth-order surface diffusion. As in thin film flows, undercompressive shocks (which violate the Lax entropy condition) can occur, and appear to dominate the dynamics when the initial surface contains slopes which are large enough. We show that when two undercompressive shocks moving in opposite directions collide, they form a shock which is undercompressive from both sides. In the height variable this is a stationary, knife-edge like ridge with steep sides; interestingly the slopes are much larger than any contained in the initial surface patterning. This suggests a possible method to pattern surfaces on the small scale more cheaply than by conventional methods: by patterning the surface initially on the macroscale to have slopes of O(1), and then bombarding it with ions, the surface may spontaneously develop features which are much smaller and steeper.

12:01PM H2.00008 The Structure of Shock Waves in Liquids

ROY BATY, CARL HAGELBERG, Los Alamos National Laboratory — This talk presents solutions of the Navier-Stokes equations that model shock waves in liquids, including water and mercury. One-dimensional jump functions are computed to describe the viscous microstructure of hydrodynamic shocks, which are not isentropic. Empirical equations of state in polynomial form are applied to the conservation laws to derive the physical flow microstructure for shock compressions and pressures up to 2.0 and 50.0 kbars, respectively. Traveling wave solutions are found for the Navier-Stokes equations by integrating the equations of motion along characteristic lines, where the shock wave jump functions are the solution of a two point boundary value problem. The empirical equations of state yield increasing shock wave jump functions for density and pressure. The solution method presented is applicable to arbitrary liquids that behave as Newtonian fluids for one-dimensional planar shock waves.

12:14PM H2.00009 Simulation of Deformation, Momentum and Energy Coupling Particles Deformed by Intense Shocks[1]

B. LIEBERTHAL, D.S. STEWART, J.B. BDZIL, U. Illinois, Urbana, IL 61801, F.M. NAJJAR, Lawrence Livermore National Laboratory, Livermore, CA 9455, S. BALACHANDAR, Y. LING, U. Florida, Gainesville, FL 32611 — Modern energetic materials have embedded solids and inerts in an explosive matrix. A detonation in condensed phase materials, generates intense shocks that deform particles as the incident shock diffracts around them. The post-shock flow generates a wake behind the particle that is influenced by the shape changes of the particle. The gasdynamic flow in the explosive products and its interaction with the deformation of the particle must be treated simultaneously. Direct numerical simulations are carried out that vary the particle-to-surrounding density and impedance ratios to consider heavier and lighter particle. The vorticity deposited on the interface due to shock interaction with the particle, the resulting particle deformation and the net momentum and energy transferred to the particle, on the acoustic and longer viscous time scale are considered. The LLNL multi-physics hydrodynamic code ALE3D is used to carry out the simulations.

[1]BL, DSS and JBB supported by AFRL/RW AF FA8651-10-1-0004 & DTRA, HDTRA1-10-1-0020 Off Campus. FMN's work supported by the U.S. DOE/ LLNL, Contract DE-AC52-07NA27344. LLNL-ABS-491794.

12:27PM H2.00010 Particle Motion Measurements Following the Impingement of a Planar Shock on a Dense Particle Field

JUSTIN WAGNER, STEVEN BERESH, SEAN KEARNEY, Sandia National Lab, BRIAN PRUETT, Sanda National Lab, ELTON WRIGHT, Sandia National Lab — The shock-induced dispersal of a dense field of particles is studied using a recently developed multiphase shock tube. The particle field is generated by a gravity-fed method that results in a spanwise curtain of 100-micron spherical particles with a volume fraction of about 20 percent. The facility drives a planar shock into the particle curtain producing interactions at incident shock Mach numbers of 1.66, 1.92, and 2.02. The gas-particle momentum transfer is calculated using a control volume technique that utilizes pressure measurements made upstream and downstream of the interaction along with the method of characteristics. High-speed schlieren imaging reveals the complex wave structure associated with the interaction and also provides a measure of the particle field motion. Particle drag estimates are made using the schlieren imaging and the control volume technique. Furthermore, the trajectories of the upstream and downstream edges of the particle field at different Mach numbers are shown to be similar when normalized by the velocity of the flow induced by the incident shock.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H3 Dynamical Systems and Chaos I 303

10:30AM H3.00001 Set-based corral control in stochastic dynamical systems: Making almost invariant sets more invariant[1]

ERIC FORGOSTON, LORA BILLINGS, PHILIP YECKO, Montclair State University, IRA SCHWARTZ, Naval Research Laboratory — We consider the problem of stochastic prediction and control in a time-dependent stochastic environment, such as the ocean, where escape from an almost invariant region occurs due to random fluctuations. We determine high-probability control-actuation sets using geometric and probabilistic methods. These methods allow us to design regions of control that provide an increase in loitering time while minimizing the amount of control actuation. Our methods provide an exponential increase in loitering times with only small changes in actuation force. The result is that the control actuation makes almost invariant sets more invariant.

[1]Research supported by the Office of Naval Research.

10:43AM H3.00002 Maximum-entropy closure for a Galerkin system of incompressible shear flow[1]

BERND R. NOACK, Institut Pprime, Poitiers, France, ROBERT K. NIVEN, The University of New South Wales at ADFA, Canberra, Australia — A statistical physics closure is proposed for Galerkin models of incompressible shear flows. This closure employs a maximum entropy (MaxEnt) principle to infer the probability distribution in Galerkin state space using exact statistical balance equations as side constraints. Application to an empirical Galerkin model of the periodic cylinder wake predicts mean amplitude values and modal energy levels in good agreement with direct numerical simulation. Recipes for more complicated Galerkin systems are provided.

[1]Partially funded by the ANR Chaires d'Excellence TUCOROM

10:56AM H3.00003 The Lagrangian description of time dependent aperiodic flows: applications in the Ocean and the Atmosphere[1]

ANA M. MANCHO, ICMAT, CSIC — Geometry has been a very useful approach for studying dynamical systems. At the basis are Poincare ideas of seeking structures on the phase space that divide it into regions corresponding to trajectories with different dynamical fates. These ideas have demonstrated to be very powerful for the description of transport in purely advective flows and important applications have been found in geophysics. However realistic flows as those obtained by altimeter satellites or from numerical simulations are highly non-periodic and to deal with these flows is a challenge because traditional methods can be used only in autonomous and time periodic dynamical systems. We describe new Lagrangian tools that are applied to general time dependent flows. In particular we discuss results on oceanic datasets taken from altimeter satellites on the Kuroshio region. We also discuss an application on reanalysis data on the Antarctic polar vortex.

[1]This research was supported by the Spanish Ministry of Science under grant MTM2008-03754, and Spanish CSIC under grant ILINK-0145. We acknowledge computational support from CESGA and CCC-UAM.

11:09AM H3.00004 Nonlinear Dynamics Analysis of a Flapping Filament in a Flowing Soap Film[1]

HAMID AIT ABDERRAHMANE, MICHAEL P. PAIDOUSSIS, McGill University, Montreal, Canada, MOHAMED FAYED, HOI DICK NG, Concordia University, Montreal, Canada — In this study we investigate the nonlinear dynamics of the flapping regime of a filament placed in a two-dimensional soap film flow for different filament lengths and flow speeds. The results show that the onset of flapping at very high Reynolds numbers is quasi-periodic, with the main flapping amplitude and frequency modulated by low-amplitude, low-frequency oscillation. At higher flow velocities, chaotic oscillation is observed. The transition to chaos occurs via the quasi-periodic route to chaos. A new bistability phenomenon is discovered in which the system alternates between the stretched-straight and oscillatory states, and is referred in this study to as "switching oscillation." Unlike some previously reported forms of bistability, in this case the system alternates between the two states continuously, without any external perturbation.

[1]This work is supported by the Natural Sciences and Engineering Research Council of Canada (NSERC).

11:22AM H3.00005 Dynamic behavior of combustion instability in a lean premixed gas-turbine combustor

HIROSHI GOTODA, TAKAYA MIYANO, Ritsumeikan University, SHIGERU TACHIBANA, Japan Aerospace Exploration Agency — Periodic and chaotic behavior in combustion dynamics that can be observed as a result of combustion instabilities in fundamental and practical combustion systems are of importance to present-day combustion physics and nonlinear science research. We experimentally investigate the dynamic behavior of the combustion instability in a lean premixed gas-turbine combustor from the viewpoint of nonlinear dynamics (Gotoda. H et al., Chaos, vol. 21, 013124, 2011). A nonlinear time series analysis clearly reveals that as the equivalence ratio increases, the dynamic behavior of the combustion instability undergoes a significant transition from stochastic fluctuation to periodic oscillation through low-dimensional chaotic oscillation. We also show that a nonlinear forecasting method is useful for predicting the short-term dynamic behavior of the combustion instability in a lean premixed gas-turbine combustor, which has not been addressed in the fields of combustion science and physics.

11:35AM H3.00006 Topological detection of Lagrangian coherent structures

JEAN-LUC THIFFEAULT[1], University of Wisconsin - Madison, MICHAEL ALLSHOUSE[2], MIT — In many applications, particularly in geophysics, we often have fluid trajectory data from floats, but little or no information about the underlying velocity field. The standard techniques for finding transport barriers, based for example on finite-time Lyapunov exponents, are then inapplicable. However, if there are invariant regions in the flow this will be reflected by a 'bunching up' of trajectories. We show that this can be detected by tools from topology. The method relies on examining a large number of topological loops, encoded symbolically. These loops wrap around the trajectories, which are viewed as topological obstacles. As the trajectories move around, they cause most loops to grow. The few loops that do not grow, or grow slowly, can be associated with coherent structures.

[1]Supported by NSF grant DMS-0806821
[2]Supported by NSF grant OCE-0645529

11:48AM H3.00007 Stochasticity enhances hydrodynamic trapping of swimming particles in chaotic flow

NIDHI KHURANA, NICHOLAS T. OUELLETTE, Yale University — We use numerical methods to investigate the dynamics of swimming particles in a two-dimensional chaotic flow field. We model the swimmers as point-like spherical particles. We include stochastic motion in addition to deterministic swimming. We find that hydrodynamic trapping, which we had previously observed for deterministic swimming alone, can be significantly enhanced by the addition of noise terms. We therefore argue that a suppression of transport due to particle activity does not depend on the detailed swimmer model, but is likely a generic effect.

12:01PM H3.00008 Dynamical systems characterization of the poor man's Navier–Stokes equations

, J.B. POLLY, City College of New York, J.M. MCDONOUGH, University of Kentucky — The Navier–Stokes (N.–S.) equations governing fluid flow consist of a system of time-dependent, multi-dimensional, non-linear partial differential equations (PDEs) which cannot be solved in real time using current, or near-term foreseeable, computing hardware. The poor man's Navier–Stokes (PMNS) equations comprise a discrete dynamical system (DDS) that is algebraic—hence, easily (and rapidly) solved—and yet which retains many (possibly all) of the temporal behaviors of the full (PDE) N.–S. system at specific spatial locations. In this investigation we outline the derivation of the PMNS equations beginning with the incompressible N.–S. equations. We then consider common techniques to understand the DDS sensitivity to initial conditions (SIC) through calculation of bifurcation diagrams, Lyapunov exponents, and fractal dimension. These techniques are studied with consideration of their ease of computation, and ability to characterize and describe system behavior. The time series generated by the DDS are used to obtain power spectral densities (PSDs) which can be used to categorize most system behaviors. Some chaotic behaviors, however, can be difficult to distinguish via PSD analysis alone; thus we investigate the ability of other methods to characterize the system response.

12:14PM H3.00009 Vortex shedding patterns, their competition, and chaos in flow past inline oscillating rectangular cylinders[1]

, SRIKANTH TOPPALADODDI, Department of Geology and Geophysics, Yale University, HARISH N. DIXIT, Mathematics Department, University of British Columbia, RAO TATAVARTI, Department of Civil Engineering, Gayatri Vidya Parishad College of Engineering, India, RAMA GOVINDARAJAN, Engineering Mechanics Unit, Jawaharlal Nehru Centre for Advanced Scientific Research, India — Different vortex shedding patterns arising in the flow past inline oscillating rectangular cylinders, at a Reynolds number of 200 is studied numerically in two-dimensions. The S-II mode of symmetric shedding, discovered in 2006, as well as the Couder-Basdevant mode [J. Fluid Mech. 173, 225-251 (1986)], seen in experiments earlier, are found numerically for the first time. Besides, a new mode of symmetric shedding, named here as S-III, is also reported. Chaotic flow in the wake of a circular cylinder, recently reported by Perdikaris *et al.* [Phys. Fluids 21(10), 101705 (2009)] is also seen in flow past the rectangular geometries here, and we show that this is indeed due to mode competition, between antisymmetric and symmetric modes of vortex shedding, in the sense of Ciliberto & Gollub [Phys. Rev. Lett. 52, 922 (1984)]. A global and reliable parameter has been constructed to "quantify" this chaos. The Lattice Boltzmann Method (LBM) has been used to solve for the flow.

[1]The Naval Physical and Oceanic Labs, Kochi are gratefully acknowledged for a grant which enabled this work to begin.

12:27PM H3.00010 Experiments on the morphology of icicles

, ANTONY SZU-HAN CHEN, STEPHEN W. MORRIS, Department of Physics, University of Toronto — Icicles form when cool water drips from an overhanging support into air whose temperature is below freezing. Ice growth is controlled by the removal of latent heat, which is transferred into the surrounding air via a thin film of water flowing over the ice surface. Predicting the shape of an icicle is a non-trivial free-boundary growth problem. The global shape emerges from the local physics of the water film, the advection-diffusion of latent heat, and the slowly evolving surface position [1]. The ice-water interface can also become unstable to form ripple patterns on the icicle surface [2]. We conducted controlled icicle experiments [3], using a table-top icicle-growing apparatus. We used image analysis to probe the evolution of both the icicle shape and the rippling instability, and we investigated their dependence on ambient temperature, water supply rate, salinity, and surface tension. Our experiments showed that under certain conditions, icicles have self-similar global profiles, but non-uniformities such as tip splitting can sometimes occur. We also found that ripple formation is correlated to the purity of water used, and the ripples climb the icicle during growth.

[1] M. B. Short, et al., Phys. Fluids 18, 083101 (2006).
[2] K. Ueno, Phys. Fluids 19, 093602 (2007).
[3] A. S. Chen, et al., Phys. Rev. E 83, 026307 (2011).

Monday, November 21, 2011 10:30AM - 12:40PM –

Session H4 Drops V: Splashing 307

10:30AM H4.00001 Drop dynamics on a thin film: Thin film rupture

, ANDREAS CARLSON, The Royal Institute of Technology, PILNAM KIM, HOWARD A. STONE, Princeton University — The spreading of a water drop on an oil film that covers a solid substrate is a common event in many industrial processes. We study in experiments the dynamics of a water drop on a thin silicone oil film and quantify its interaction with the solid substrate that supports the film. The oil film becomes unstable and ruptures for solids that are hydrophilic. We determine the "waiting time," the time it takes the water drop to drain the silicone film. This timescale is found to highly depend on how well water wets the solid, illustrating the interplay between intermolecular and hydrodynamic forces in the phenomenon. A phase diagram for the thin film stability is extracted based on waters equilibrium contact angle on the solid, which shows that we can either promote or inhibit de-wetting. As water comes in direct contact with the solid, it spreads and peels off the silicone film. We show the influence of viscosity, equilibrium contact angle and film height on the opening radius of the hole formed as the solid de-wets.

10:43AM H4.00002 High-speed microdroplet impact on smooth solid surfaces

, CLAAS WILLEM VISSER, YOSHIYUKI TAGAWA, NIKOLAI OUDALOV, CHAO SUN, DETLEF LOHSE, University of Twente — We present experimental observations of high-speed microdroplet impact on smooth solid surfaces (roughness $R_{RMS} \approx 1\text{nm}$), using high-speed imaging up to 1 million frames per second. The ranges of the droplet size and velocity are $\mathcal{O}(0.01)$ - $\mathcal{O}(0.1)$ mm and $\mathcal{O}(1)$- $\mathcal{O}(100)$ m/s, respectively. Droplet generation is achieved by break-up of a high-velocity microjet. The final droplet radius after impact and the development of droplet radius in time are investigated for impact of the first droplet of the droplet train. The results are compared with models available in literature.

10:56AM H4.00003 Splash control of drop impacts using small geometric targets

, GABRIEL JUAREZ, PAULO E. ARRATIA, University of Pennsylvania — We present the results of an experimental investigation on the impact of droplets with solid planar surfaces that are of the same dimension as the drop diameter. This experimental configuration allows us to examine the splashing process as governed only by inertial and capillary forces. The cross sectional areas of the targets are regular polygons and the number of sides (N) varies from 3 to 10. Splashing results for polygon targets are compared to splashing on a cylindrical post, the limiting case of infinite sides. Upon impact, we observe that the splash radius increases exponentially in time. We also find that for particular geometrical shapes ($N < 8$), it is possible to influence splashing process. The resulting splash resembles the shape of the target with a rotation equal to $180/N$. The breakup of the lamella into secondary jets is accurately controlled and equal to the number of vertices. For targets with more than 8 sides, the splash becomes irregular and does depend on target shape.

11:09AM H4.00004 ABSTRACT WITHDRAWN —

11:22AM H4.00005 Influence of gas properties in drop splashing

, CHRISTOPHE JOSSERAND, ZHEN JIAN, Institut D'Alembert, CNRS & UPMC, STEPHANE POPINET, NIWA, PASCAL RAY, STEPHANE ZALESKI, Institut D'Alembert, CNRS & UPMC, FCIH COLLABORATION — We study numerically the influence of the surrounding gas in the drop impact dynamics. We observe that when super-hydrophobic boundary conditions are taken, splashing is always present, that we explain through a lubrication argument. On the other hand, when partial wetting is allowed, a splashing/spreading transition is observed. This transition is strongly influenced by the viscosity ratio but shows also a slight dependence with the gas pressure in qualitative agreement with experimental results.

11:35AM H4.00006 Splashing or not , JULIE ALBAGNAC, Brown University, JOHN KOLINSKI, SHMUEL RUBINSTEIN, Harvard University, SHREYAS MANDRE, Brown University — The splashing of a droplet when impacting a solid surface is common to our everyday experience as well as to industrial applications that require controlled deposition of liquid mass. Still the mechanism for splashing is not well understood. A recent study showed that a decrease in the ambient pressure inhibits splashing, motivating a hypothesis on the existence of a thin film of air trapped between the drop and the surface. The early dynamics of splashing could occur while the drop is still spreading on an air film. To gain insight into this early dynamics, we supplement the side view with a synchronized bottom view, obtained using a novel Total Internal Reflection technique. I will discuss the existence of a transition regime between spreading and splashing. This regime appears by changing the impact velocity or the ambient pressure, while keeping the other fixed.

11:48AM H4.00007 Effect of viscosity on dynamics of drop impacting solid surface , RAVI SINGH, Brown University, SHREYAS MANDRE, Brown University — High velocity impact between liquid drop and solid surface produces a splash via ejecting a thin sheet of liquid near the impact point. Recent study suggests that the drainage of an intervening air film mediate the origination of this liquid sheet. The nature of this drainage process depends on the viscosity of the drop liquid. Most notably, for vanishing liquid viscosity, the air film drains in a finite time via developing a singularity in finite time where curvature, velocity and pressure diverges near point of contact. On the other hand, in the limit of large liquid viscosity this drainage takes an infinite amount of time. But real liquids have finite viscosity. What is the nature of gas drainage for real liquid drops and how does viscosity affect this drainage? This motivated us to do theoretical and numerical study of the drainage process including both the liquid inertia and viscosity. In particular, we investigate the effect of a finite viscosity on the finite-time singularity that develops near the point of contact.

12:01PM H4.00008 Air cushioning in drop impact , JOLET DE RUITER, JUNG OH, DIRK VAN DEN ENDE, FRIEDER MUGELE, University of Twente — Liquid drops impacting on solid surfaces deform under the influence of the ambient gas that needs to be squeezed out before a true solid-liquid contact can be established. We demonstrate experimentally the existence of this theoretically predicted air layer and follow its evolution with time for moderate impact speeds ($We \sim 1 \ldots\ 10$) using reflection interference microscopy with a thickness resolution of approximately 10nm. For a wide range of fluid properties (ρ, γ, η) we find a very robust generic behavior that includes the predicted formation of a dimple in the center of the drop with a local minimum of the air film thickness at its boundary. Depending on *We* as well as the fluid properties, a skating layer of more or less constant thickness as well as a second local minimum of the air film thickness farther away from the drop center develop in time. Eventually, solid-liquid contact is generated via random nucleation event. The nucleation spot spreads across the drop-substrate interface within a few milliseconds. This process can lead to the entrapment of an air bubble.

12:14PM H4.00009 The influence of the surrounding gas on drop impact onto a wet substrate , ROBERT DEEGAN, LI ZHANG, JAMESON TOOLE, University of Michigan — The impact of a droplet with a wet or solid substrate creates a spray of secondary droplets. The effect of the surrounding gas on this process was widely neglected prior to the work of Xu, Zhang, & Nagel which showed that lowering the gas pressure suppresses splashing for impact with a dry solid substrate. Here we present the results of our experimental investigation of the effect of the surrounding gas on the evolution of splashes from a wet substrate. We varied the density and pressure of the surrounding gas. We find quantitative changes to the onset thresholds of splashing and on the size distribution of, but no qualitative changes. The effects are most pronounced on the evolution of the ejecta sheet.

12:27PM H4.00010 Drop impact dynamics of complex fluids on dry, nanotextured surfaces , KYOO-CHUL PARK, VIVEK SHARMA, Dept of Mechanical Eng., MIT, ROBERT E. COHEN, Dept of Chemical Eng., MIT, GARETH H. MCKINLEY, Dept of Mechanical Eng., MIT — The deposition of aqueous drops on non-wetting surfaces is an important problem for many applications, including spraying of pesticides and herbicides onto plant leaves. The addition of a small amount of high molecular weight polymer has been demonstrated to suppress drop rebound. The high extensional viscosity of polymer solutions and increased dissipation in polymer solutions near the receding contact line are cited as two distinct antirebound mechanisms. Using drop impact experiments on both natural and synthetic micro- and nano-textured surfaces with controlled wetting characteristics we examine the role of viscosity, surface tension, elasticity and inertia on expansion, retraction, and rebound of well-characterized viscoelastic fluids. By varying the radius of gyration of the polymer coils in solution as well as texture length scale, we can achieve complete drop rebound on nanotextured surfaces even for high molecular weight polymer solutions. By contrast, in similar conditions on natural microtextured surfaces complete adhesion is observed.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H5 CFD IV: Numerical Methods I 308

10:30AM H5.00001 A Simple and Efficient Parallel Implementation of the Fast Marching Method[1] , JIANMING YANG, FREDERICK STERN, University of Iowa — The fast marching method is a widely used numerical method for solving the Eikonal equation arising from a variety of applications. However, this method is inherently serial and doesn't lend itself to a straightforward parallelization. In this study, we present a simple and efficient algorithm for the parallel implementation of the fast marching method using a domain decomposition approach. Properties of the Eikonal equation are explored to greatly relax the serial interdependence of neighboring sub-domains. Overlapping sub-domains are employed to reduce communication overhead and improve parallelism among sub-domains. There are no iterative procedures or rollback operations involved in the present algorithm and the changes to the serial version of the fast marching method are minimized. Examples are performed to demonstrate the efficiency of our parallel fast marching method.

[1]This study was supported by ONR.

10:43AM H5.00002 ABSTRACT WITHDRAWN –

10:56AM H5.00003 A new fully explicit algorithm for incompressible flows , SANGRO PARK, CHANGHOON LEE, Yonsei University — The Poisson equation for pressure arising from nonzero divergence of the nonlinear term in the integration of the Navier-Stokes equations requires a lot of computational cost except for cases with periodic domain. In order to mitigate this cost, we propose a new project algorithm which is fully explicit, thus not requiring iterations. The projection operator, $1 - \kappa_i\kappa_j/\kappa^2$, which projects any vector field with divergence into the divergence-free subspace in the Fourier space, when transformed into the physical space, shows decaying distribution with the distance from the point in question. This allows truncation so that the resulting local distribution of the projection operator, through convolution, can be used to obtain projected nonlinear terms which has relatively small divergence. This "approximate" projection scheme was then applied to direct numerical simulation of isotropic turbulence to investigate effectiveness and efficiency of the scheme in reducing divergence and correct projection of the nonlinear terms through the statistical properties of the turbulent flow. Performance of the scheme in a variety of aspects is investigated and details will be presented in the meeting.

11:09AM H5.00004 A coarse-grid projection method for accelerating incompressible flow computations , OMER SAN, ANNE STAPLES, Virginia Tech — We present a coarse-grid projection (CGP) algorithm for accelerating incompressible flow computations, which is applicable to methods involving Poisson equations as incompressibility constraints. CGP methodology is a modular approach that facilitates data transfer with simple interpolations and uses black-box solvers for the Poisson and advection-diffusion equations in the flow solver. Here, we investigate a particular CGP method for the vorticity-stream function formulation that uses the full weighting operation for mapping from fine to coarse grids, the third-order Runge-Kutta method for time stepping, and finite differences for the spatial discretization. After solving the Poisson equation on a coarsened grid, bilinear interpolation is used to obtain the fine data for consequent time stepping on the full grid. We compute several benchmark flows: the Taylor-Green vortex, a vortex pair merging, a double shear layer, decaying turbulence and the Taylor-Green vortex on a distorted grid. In all cases we use either FFT-based or V-cycle multigrid linear-cost Poisson solvers. Reducing the number of degrees of freedom of the Poisson solver by powers of two accelerates these computations while, for the first level of coarsening, retaining the same level of accuracy in the fine resolution vorticity field.

11:22AM H5.00005 Explicit Multirate Runge-Kutta Time Advancement for Navier-Stokes on Unstructured Grids[1] , PAUL COVINGTON, FRANK HAM, PARVIZ MOIN, CTR, Stanford University — Unstructured grids of complex geometries often contain a handful of very small or poor quality elements that severely limit the global CFL-restricted timestep for explicit time advancement. So-called Multirate Runge-Kutta (MRK) methods attempt to overcome this limitation by using different time integration schemes in different regions of the domain based on a local stability criterion, thereby reducing overall computational cost (E. Constantinescu, JSC 2007 & M. Schlegel, JCAM 2009). In this study, concepts from MRK are combined in a novel way to speedup an unstructured compressible finite volume code. As a validation case, the evolution of a 2D Euler vortex on a recursively refined grid will be presented. Algorithmic issues such as load balancing will also be discussed since these are crucial to approaching the theoretically achievable speedup. The practical implications of this technique will be demonstrated on a large-scale simulation of a low Mach number automotive fan.

[1]Work supported by Robert Bosch, LLC.

11:35AM H5.00006 Numerical simulation, prediction and experimental control of the dynamic behavior of a rotating magnetic particle chain , YANG GAO, Eindhoven University of Technology, TAE GON KANG, School of Aerospace and Mechanical Engineering, MARTIEN HULSEN, Eindhoven University of Technology, JAAP DEN TOONDER, Eindhoven University of Technology and Royal Philips Electronics — A simple and fast numerical method is developed capable of accurately determining the 3D rotational dynamics of magnetic particle chains in an infinite fluid domain. The focus is to control the alternating breakup and reformation of the bead chains which we believe is essential to achieve effective fluid mixing at small scales. The numerical scheme makes use of both the magnetic and hydrodynamic interactions between the particles. It is shown that the inclusion of hydrodynamic interaction between the particles is crucial to obtain a true description of the particle dynamics. A small error of deviation is observed when benchmarking the numerical scheme against the direct simulation method. The numerical results are compared with experiments showing good agreement both qualitatively and quantitatively. In addition, a dimensionless number is derived as the sole control parameter for the rotational bead chain dynamics.

11:48AM H5.00007 One Possible Pitfall with Current Practice in Grid-Convergence , RYAN Z. DAVIS, RICHARD SKIFTON, AKIRA TOKUHIRO, University of Idaho — It is claimed that a possible flaw could exist in common-practice grid convergence studies. The problem comes when a field point value is observed near a discontinuity or a strong gradient and used to calculate the order-of-convergence, or even confirm grid convergence. A discrete domain can produce a "smoothing" effect in the vicinity of strong gradients. When field point values are observed near these areas their values can change dramatically, or not at all, depending on where the discrete points lie. This can lead the user to believe grid convergence has been achieved when in reality it has not. A submerged jet is used as a case study to demonstrate how the order-of-convergence can be affected when observing field point values near and far from strong gradients.

12:01PM H5.00008 Toward 3D vortex methods with deforming basis functions , LOUIS ROSSI, CLAUDIO TORRES, University of Delaware — We present recent results extending methods for 2D vortex methods with deforming basis functions to three dimensions. Vortex methods are numerical schemes for approximating solutions to the Navier- Stokes equations using a linear combination of moving basis functions to approximate the vorticity field of a fluid. Typically, the basis function velocity is determined through a Biot-Savart integral applied at the basis function centroid. One outcome of rigorous analysis is an new naturally adaptive high order 2D method with elliptical Gaussian basis functions that deform as they move according to flow properties. This new class of methods is very unusual because the basis functions do not move with the physical flow velocity at the basis function centroid as is usually specified in vortex methods. The resulting analysis leads to deforming, ellipsoidal basis functions capable of achieving high spatial order. We now extend these results to three dimensions where traditional vortex methods suffer the additional shortcoming of not preserving the divergence of the vorticity field. We will discuss the latest results on our efforts to develop a complete 3D vortex method with adaptive, deforming blobs.

12:14PM H5.00009 Fully explicit algorithms for fluid simulation , JONATHAN CLAUSEN, Sandia National Laboratories — Computing hardware is trending towards distributed, massively parallel architectures in order to achieve high computational throughput. For example, Intrepid at Argonne uses 163,840 cores, and next generation machines, such as Sequoia at Lawrence Livermore, will use over one million cores. Harnessing the increasingly parallel nature of computational resources will require algorithms that scale efficiently on these architectures. The advent of GPU-based computation will serve to accelerate this behavior, as a single GPU contains hundreds of processor "cores." Explicit algorithms avoid the communication associated with a linear solve, thus parallel scalability of these algorithms is typically high. This work will explore the efficiency and accuracy of three explicit solution methodologies for the Navier–Stokes equations: traditional artificial compressibility schemes, the lattice-Boltzmann method, and the recently proposed kinetically reduced local Navier–Stokes equations [Borok, Ansumali, and Karlin (2007)]. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

12:27PM H5.00010 Analysis of boundary conditions and diffusion schemes for SPH methods , ZHENYU HE, LOUIS ROSSI, University of Delaware — In this paper, we systematically explore the accuracy and stability of different methods for satisfying boundary conditions and capturing viscous diffusion in smoothed particle hydrodynamics (SPH). Smoothed particle hydrodynamics (SPH) is a Lagrangian method for compressible and incompressible flows. The state of fluid system is represented by a set of moving basis functions which interpolate the material properties. The mesh-free formulation of the method and its inherent stability make it popular for problems that have complex geometry or large deformations. Our research focuses mathematically on an accurate and efficient treatment for physical boundary conditions. Also, we analyze several smoothing kernels and diffusion schemes. We compare different techniques for non-slip, non-penetration conditions such as fixed fluid particles, ghost particles and boundary particle forces. We verify our results by comparing computations to exact solutions for 2D planar and circular, steady and unsteady Couette flows.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H6 Drops VI: Thermal Effects 309

10:30AM H6.00001 Drop impact on superheated surfaces , TUAN TRAN, ERIK-JAN STAAT, ANDREA PROSPERETTI, CHAO SUN, DETLEF LOHSE, PHYSICS OF FLUIDS, UNIVERSITY OF TWENTE TEAM — We let millimeter droplets impact on superheated smooth surfaces. Depending on the impact velocity and the surface temperature, the droplet either immediately boils under spray formation when it contacts the surface ("wet boiling"), or without any surface contact forms a Leidenfrost vapor layer towards the hot surface and bounces back ("gentle film boiling"), or even breaks up in the middle during that process so that it basically immediately vaporizes ("violent film boiling"). We experimentally determine the phase diagram and qualitatively account for the transitions between the different regimes.

10:43AM H6.00002 Boiling water droplet behavior impinging on heated liquid metal surface , TETSU YASUI, KOJI OKAMOTO, SATOSHI SOMEYA, the University of Tokyo — We observed boiling behavior of a water drop impinging on a heated liquid metal surface. By using liquid metal as impinged surface, we are able to observe boiling behavior of a water drop impinging on perfectly smooth surface. As a result of experiment, we found that a water drop starts boiling with the time delay after impinging and it changes by Weber number and initial surface temperature. And we also found that there are three factors which induce the direct contact of a water drop and surface by shooting in a high frame rate.

10:56AM H6.00003 Manipulating Leidenfrost temperature with surface texture , HYUK-MIN KWON, JAMES BIRD, KRIPA VARANASI, Massachusetts Institute of Technology — When a drop contacts a surface that is at a sufficiently high temperature, the drop can float on it's own vapor in a process known as the Leidenfrost effect. Although it is known that the critical temperature needed to achieve this effect depends on the properties of the drop and its vapor, often these parameters are fixed for a particular process. Here, we demonstrate a new way to control the critical temperature through the surface structure.

11:09AM H6.00004 Viscous mechanism for Leidenfrost propulsion on a ratchet , GUILLAUME DUPEUX, CHRISTOPHE CLANET, Physique et Mecanique des Milieux Heterogenes, CNRS, ESPCI, Paris France & Ladhyx, CNRS, Ecole Polytechnique, Palaiseau, France, STEFFEN HARDT, Center of Smart Interfaces, Technische Universitat Darmstadt, Germany, DAVID QUERE, Physique et Mecanique des Milieux Heterogenes, CNRS, ESPCI, Paris France & Ladhyx, CNRS, Ecole Polytechnique, Palaiseau, France — An evaporating drop placed on a ratchet shaped substrate self-propels, as discovered by Linke et al. in 2006. Sublimating platelets of dry ice do the same, and we discuss a possible viscous mechanism for these motions. We report that the flow of vapor below the levitating material is rectified by the asymmetric teeth of the ratchet, in the direction of descending slopes along each tooth. As a consequence, the resulting viscous stress can entrain the material in the same direction, and we discuss the resulting self-propelling force.

11:22AM H6.00005 Flow near the contact line of an evaporating drop , HANNEKE GELDERBLOM, OSCAR BLOEMEN, ÁLVARO G. MARíN, DETLEF LOHSE, JACCO H. SNOEIJER, Physics of Fluids, University of Twente — Evaporation of water droplets with a pinned contact line is investigated in the entire range of possible contact angles (up to 150°). We measured the evolution of the mass and the contact angle of droplets on superhydrophobic carbon nanofiber substrates. The experimental data are in quantitative agreement with a diffusion-based model, and can be collapsed onto a single universal curve for all droplet sizes and initial contact angles. However, the nature of the flow inside the drop near the contact line has remained unclear: it has been argued that the lubrication approximation does not apply in the contact-line region, and that the Marangoni effect can overcome the evaporation-driven outward flow. Here, we resolve these questions by analytically solving the Stokes flow field in a wedge geometry, imposing the evaporative flux as a boundary condition.

11:35AM H6.00006 Diffusion-Controlled Evaporating Perfectly wetting meniscus , JEAN-PIERRE NJANTE, STEPHEN MORRIS, UC Berkeley — The Stefan diffusion theory is often used to predict the evaporation rate from capillaries. Following Derjaguin et al (Bull. Rilem, 1965), we show here that because the simple diffusion theory does not include mass loss from the precursor film, it significantly under-estimates mass loss from small (μm-sized) capillaries. We make the following simplifying assumptions. (a) Diffusion is rate limiting; as a result, liquid and vapor at the interface are in local thermodynamic equilibrium. (b) The system is effectively isothermal; though evaporation induces liquid temperature differences, they are kinetically negligible. (c) Transport of the evaporating molecules is by axial-diffusion only. At a given axial location x, the total mass flow $\dot{m}$ is the sum of the flow within the thin liquid film and that occurring by diffusion within the gas $-\dot{m} = \frac{h^3}{3\nu}\frac{dp_\ell}{dx} + \frac{\rho_s D}{\rho R T_w}(a-h)\frac{dp_\ell}{dx} = constant$ where ν is the kinematic viscosity, R the gas constant, D the diffusion coefficient, ρ the liquid density, T_w the wall temperature, $2a$ the channel gap thickness, and ρ_s the saturation density. The interface shape $h(x)$ is related to the liquid pressure at the interface $p_\ell(x)$ by the normal stress equation $P_a - p_\ell = \gamma\frac{d^2h}{dx^2} + \frac{A}{h^3}$ where P_a is the total gas pressure and A the Hammaker constant. Our model differs from that of Derjaguin et al because including surface tension γ allows us to calculate the shape of the whole meniscus, and so to relate our results for mass loss to that given by standard theory.

11:48AM H6.00007 Planar Jumping-Drop Thermal Diodes , JONATHAN BOREYKO, YUEJUN ZHAO, CHUAN-HUA CHEN, Duke University — Phase-change thermal diodes transport heat asymmetrically with a large rectification coefficient unmatched by their solid-state counterparts, but are limited by either the gravitational orientation or one-dimensional configuration. We report a planar phase-change diode scalable to large areas with an orientation-independent diodicity of up to 100, in which water/vapor is enclosed by parallel superhydrophobic and superhydrophilic plates. The thermal rectification is enabled by spontaneously jumping dropwise condensate which only occurs when the superhydrophobic surface is colder than the superhydrophilic surface. Our jumping-drop thermal diode is expected to be particularly useful for the thermal protection of planar electronic components and the thermal regulation of large-area energy harvesting systems.

12:01PM H6.00008 Droplet manipulation using light-induced thermal flows on an amorphous silicon thin film , HORIM LEE, JIN SUNG YOON, KWAN HYOUNG KANG, Department of Mechanical Engineering, POSTECH — We present a droplet-manipulation method using light-induced thermal flows in oils. The flows are originated from Marangoni and buoyancy effects due to temperature gradient, generated by the adsorption of light in an amorphous silicon thin film. The effects of the types and thickness of the oil on the flows are explored experimentally and numerically. Using this method, we can manipulate a droplet including cells in an extremely simple system without the damage on cell viability.

12:14PM H6.00009 Pattern Formation in Drying Drops of Polystyrene/Water nanofluids , DAVID BRUTIN, BENJAMIN SOBAC, Aix-Marseille University - IUSTI — We study the pattern formation and the evaporation dynamics of drying drops of polystyrene/water based nanofluids with concentrations ranging from 0.01% to 6%. Cracks formation is evidenced to depend on the nanoparticles concentration. The dynamics of evaporation is recorded using an electronic balance with an accuracy of 10μg. A top view recording enables to analyze the pattern formation in relation with the mass evolution. We determine several key parameters such as the time of evaporation, the wetting diameter, the final solid deposition diameter, the number and the spacing of the cracks. We evidence a ring formation above a critical concentration. We evidenced by change of the surrounding humidity in the range of 10 to 90% that this pattern remains constant. The pattern formation is influenced by the liquid phase evaporation dynamics but only depends on the concentration in nanoparticles. These results are of great interest regarding the formation of droplets in several areas such as inkjet printing, pharmacology...

12:27PM H6.00010 Numerical study of thermocapillary instabilities in evaporating annular pools and sessile droplets[1] , PEDRO J. SÁENZ, PRASHANT VALLURI, KHELLIL SEFIANE, University of Edinburgh, GEORGE KARAPETSAS, OMAR K. MATAR, Imperial College London — We investigate thermocapillary flows due to temperature-induced surface tension gradients in annular liquid pools via full two-phase direct numerical simulations in 3D. Phase-change, interface deformation and wettability phenomena are taken into consideration by using a variant of the volume-of-fluid method. The simulation results are validated against experiments (Schwabe et al. 2003 & Riley et al. 1998) and theory (Smith & Davis 1983). The transient results show the evolution of the flow towards an oscillatory state characterized by interfacial hydrothermal waves (HTWs). We present the effects of non-uniform evaporation fluxes and the liquid depths on the linear and non-linear development of these thermocapillary instabilities. The influence on bulk flows, surface temperature patterns and interface deformations are also shown. We finally introduce spontaneously self-excited HTWs in evaporating sessile droplets simulated using novel numerical methods and compare the results against analytical models and experiments.

[1]Fundación Caja Madrid & EPSRC DTA

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H7 Turbulent Boundary Layers V 310

10:30AM H7.00001 Organization of very-long coherent structures in a turbulent pipe DNS with long domain[1] , RONALD ADRIAN, JON BALTZER, Arizona State University, XIAOHUA WU, Royal Military College of Canada — Regions of negative velocity fluctuation are extracted from a turbulent pipe simulation at $Re_\tau = 685$. The lengths of these regions are characterized for a range of thresholds. The organizational patterns of these structures are analyzed using a linear stochastic estimate of a negative velocity fluctuation event. Instantaneous examples of these events are compared with the patterns in the estimate to understand the relationship between structure organization and the two-point spatial correlation. Close examination of the very long motions reveal distinct forms of smaller motions with different characteristic scales. Studying these structural elements provides insight into how the smaller motions compose the very long motions. Cross-sections in various planes are compared to features observed in experimental investigations, including vortical motions. Three-dimensional vortical structures are extracted using several methods, and their geometries and relationships with the velocity motions are examined.

[1]Supported by NSF Award CBET-0933848.

10:43AM H7.00002 Properties of Wide Spanwise or Azimuthal Velocity Modes in Turbulent Boundary Layers, Channels, and Pipes[1] , JON BALTZER, RONALD ADRIAN, Arizona State University — While the motions that exist after instantaneously spanwise averaging a plane turbulent channel or boundary layer would vanish as spanwise domain size is increased, the motions from azimuthally averaging a pipe have clearer physical significance due to the constrained geometry. These averages also arise as the spanwise $k_z = 0$ and azimuthal $k_\theta = 0$ modes in a Fourier (or POD) decomposition. The $k_z/k_\theta = 0$ modes of velocity contain 3.0, 3.5, and 7.2% of the streamwise, wall-normal, and spanwise turbulent kinetic energy at $0.25h$ above the wall for a $Re_\tau = 590$ channel DNS with πh wide domain (Moser, Kim, & Mansour, Phys. Fluids, 1999), and 1.5, 2.3, and 4.2% for a $Re_\tau = 685$ pipe DNS at $0.25R$ above the wall. These increase to 9–11% for each component at the channel centerline. These modes are studied for channel and boundary layer DNSs with several domain widths and a pipe. Time evolution and traveling wave behavior of these modes are explored. The modes are characterized with proper orthogonal decomposition, and the results are compared to 2D POD modes for planes without spanwise/azimuthal averaging.

[1]Supported by NSF Award CBET-0933848.

10:56AM H7.00003 Large-scale coherent motion in turbulent pipe flow , JERRY WESTERWEEL, TU Delft, SEBASTIAN GROSSE — Fully-developed turbulent pipe flow at bulk Reynolds numbers ranging from Re = 10 000 to 44 000 has been investigated experimentally using high-speed PIV in a plane perpendicular to the mean flow. A stereoscopic setup was used to enable the reconstruction of all three components of the entire azimuthal velocity field. The application of Taylor's hypothesis allowed to reconstruct the quasi-instantaneous streamwise extension of the flow field. Individual recording sequences cover more than 150 bulk scales based on the bulk velocity and the pipe radius such that even the largest expected streamwise extends of the flow structures are captured. The azimuthal flow field scaling was found to be consistent with results reported in previous studies. A steep decrease of azimuthal width towards values found in turbulent boundary layers could be observed at low Reynolds number indicating the near-wall region to be unaffected by the confinement of the pipe geometry. At wall distances $0.5 < y/R < 0.8$ azimuthal growth nearly stagnates. The large-scale coherent flow field at different wall-normal positions exhibits a very high instantaneous similarity. Streak-width distributions exhibit a flattening of the present azimuthal streak extensions throughout the logarithmic and wake region.

11:09AM H7.00004 Time-resolved PIV in fully developed turbulent pipe flow[1] , LEO HELLSTRÖM, ALEXANDER SMITS, Princeton University — Stereoscopic particle image velocimetry was used to study the three-component velocity field in fully developed turbulent pipe flow, to investigate the structure and behavior of the large and very large scale motions in the outer layer. The data was acquired with a high speed camera, making it possible to resolve the velocity field in time for Reynolds numbers ranging from 1.3×10^4 to 3.6×10^4. Proper Orthogonal Decomposition was performed on the data to extract the most energetic modes in the flow, which are believed to correspond to the very large scale motions. We show that a small number of modes may be used to reconstruct these structures. The procedure can be used as a VLSM filter to further investigate their behavior.

[1]Supported under ONR Grant N00014-09-1-0263 and NSF Grant CBET-1064257.

11:22AM H7.00005 On the generation and growth of hairpin vortex packets in wall turbulence[1] , BEVERLEY MCKEON, California Institute of Technology, ATI SHARMA, University of Sheffield — We demonstrate that the critical layer framework of McKeon & Sharma (JFM, 2010) which was previously shown to capture aspects of the statistics of wall turbulence, also gives insight into the appearance and organization of vortical structure. The model reconciles aspects of the statistical and structural pictures of wall turbulence. Choosing suitable scales, the superposition of left- and right-going propagating modes which are attached to the wall reproduces the development of a periodic array of pro- and retro-grade hairpin vortices. The addition of mean shear associated with the turbulent mean velocity profile suppresses the retrograde vortices and enhances the prograde vortices. Further, superposition of attached modes with differing convection velocities produces packets of hairpins that evolve in both space and time. Characteristics of these vortical structures and a comparison with the vortex signatures observed in experiments will be described.

[1]The support of AFOSR under grant FA 9550-09-1-0701 is gratefully acknowledged.

11:35AM H7.00006 The lifetime and evolution of vortical structures in the turbulent boundary layer , JEFFREY LEHEW, California Institute of Technology, MICHELE GUALA, St. Anthony Falls Laboratory and Dep. Civil engineering, UMN, BEVERLEY MCKEON, California Institute of Technology — The evolution and persistence of vortices in wall normal and wall parallel planes of a zero-pressure gradient turbulent boundary layer are investigated using time resolved 2D-2C particle image velocimetry at a moderate Reynolds number ($Re_\tau = 470$). Measurements are performed with a large field of view: $5\delta \times 1.2\delta$ (streamwise x wall normal) in the wall normal plane and both $10\delta \times 5\delta$ and $4.3\delta \times 2.2\delta$ (streamwise x spanwise) in the wall parallel planes. Given that the data provides information in both the streamwise direction and time simultaneously, not only can the vortices be tracked as they convect downstream, but the evolution of the surrounding flow structures can be studied in the reference frame of the vortex. In addition to investigating statistics associated with the lifetime and convection velocity of vortices at various wall normal locations, the effects of spatial and/or temporal filters on the identification and shape of vortices will be investigated and an interpretation of vortices in such a filtered velocity field will be given. Support for this work from the AFOSR under award# FA9550-09-1-0701 is gratefully acknowledged.

11:48AM H7.00007 Convection Velocities of Coherent Structures in a Thermally Perturbed Turbulent Boundary Layer , REBECCA ROUGHT, California Institute of Technology, STANISLAS GORDEYEV, University of Notre Dame, BEVERLEY MCKEON, California Institute of Technology — Increasing use of airborne laser-based communication systems has led to a need to understand the effects of turbulent boundary layers on the performance of such systems. As a beam of coherent light passes through a variable density flow, it is degraded as a result of changes in the index of refraction associated with the density fluctuations. In order to study this phenomenon, a zero-pressure gradient flat plate is subjected to a thermal perturbation, causing an internal, heated layer to form within the boundary layer. A Malley probe is used downstream of the perturbation to measure the Optical Path Difference of a laser traversing the flow. This probe consists of two parallel laser beams which deflect as heated turbulent structures convect through the beams. Cross-correlating the deflection angle spectrum of the two beams provides convective velocity information. The mean convective velocity is compared to the free stream velocity and estimates of structure convection velocities in the literature obtained using other diagnostics.

12:01PM H7.00008 Very large-scale motions in a turbulent pipe flow[1] , JAE HWA LEE, SEONG JAE JANG, HYUNG JIN SUNG, KAIST — Direct numerical simulation of a turbulent pipe flow with Re_D=35000 was performed to investigate the spatially coherent structures associated with very large-scale motions. The corresponding friction Reynolds number, based on pipe radius R, is R^+=934, and the computational domain length is $30R$. The computed mean flow statistics agree well with previous DNS data at Re_D=44000 and 24000. Inspection of the instantaneous fields and two-point correlation of the streamwise velocity fluctuations showed that the very long meandering motions exceeding 25R exist in logarithmic and wake regions, and the streamwise length scale is almost linearly increased up to y/R$\sim$0.3, while the structures in the turbulent boundary layer only reach up to the edge of the log-layer. Time-resolved instantaneous fields revealed that the hairpin packet-like structures grow with continuous stretching along the streamwise direction and create the very large-scale structures with meandering in the spanwise direction, consistent with the previous conceptual model of Kim & Adrian (1999).

[1]This work was supported by the Creative Research Initiatives of NRF/MEST of Korea (No. 2011-0000423).

12:14PM H7.00009 Why do large and small scales couple in a turbulent boundary layer? , PROMODE R. BANDYOPADHYAY, Naval Undersea Warfare Center, Newport, RI — Correlation measurement, which is not definitive, suggests that large and small scales in a turbulent boundary layer (TBL) couple. A TBL is modeled as a jungle of interacting nonlinear oscillators to explore the origin of the coupling. These oscillators have the inherent property of self-sustainability, disturbance rejection, and of self-referential phase reset whereby several oscillators can phase align (or have constant phase difference between them) when an "external" impulse is applied. Consequently, these properties of a TBL are accounted for: self-sustainability, return of the wake component after a disturbance is removed, and the formation of the 18o large structures, which are composed of a sequential train of hairpin vortices. The nonlinear ordinary differential equations of the oscillators are solved using an analog circuit for rapid solution. The post-bifurcation limit cycles are determined. A small scale and a large scale are akin to two different oscillators. The state variables from the two disparate interacting oscillators are shown to couple and the small scales appear at certain regions of the phase of the large scale. The coupling is a consequence of the nonlinear oscillatory behavior. Although state planes exist where the disparate scales appear de-superposed, all scales in a TBL are in fact coupled and they cannot be monochromatically isolated.

12:27PM H7.00010 Behavior captured by Lagrangian coherent structures near the wall in a turbulent boundary layer , ZACHARY WILSON, Portland State University (PSU), MURAT TUTKUN, Norwegian Defence Research Establishment, RAÚL BAYOÁN CAL, PSU — In this study, Lagrangian coherent structures (LCS) are identified in a plane that is 50 viscous lengths from the wall in a turbulent boundary layer. To locate the LCS, the criteria introduced by Haller (2011)[1], which is based on a variational approach to optimizing normal repulsion (or attraction), is applied. The flow map is constructed by using time resolved velocity fields, obtained from high speed (1.5 kHz) PIV measurements, to compute trajectories (Integration time $\sim y^+\delta_\nu/u_\tau$ and time steps $\sim \delta_\nu/u_\tau$). The various turbulent behaviors captured by the repelling and attracting LCS are examined. In particular, the computed LCS are compared with plots of streamwise momentum, velocity fluctuations, dissipation, and various Eulerian fields that highlight vortical structures. It is found that different LCS depict structures corresponding to different Eulerian criteria. For instance, some LCS are boundaries between high and low momentum regions, while others surround vortical structures and/or exhibit high dissipation.

[1]Haller, G. Physica. D 240 7. (2011)

Monday, November 21, 2011 10:30AM - 12:40PM – Session H9 Industrial Applications 312

10:30AM H9.00001 Investigations of the Effects of Distortion on Trajectory of Diesel Particulate Matter (PM)[1] , HAMID RAHAI[2], CEERS/California State University, Long Beach, EHSAN SHAMLOO TEAM[3] — Exposures to diesel PM within urban areas have resulted in elevated respiratory illnesses and risk of premature cardiac death. The present investigation is part of our continuous efforts to understand the relationship between diesel exhausts concentrations and local urban aerodynamics, in metropolitan areas where significant diesel vehicle activities are present. Wind tunnel experiments and field tests were performed to understand the effects of distortion caused by building structures on trajectory of the diesel PM, emitted from diesel engine exhausts in cross flows. Our previous similar investigations without distortion have shown a linear decay in PM concentration with a 10 percent slope. However, when the structures were present, PM concentration increased significantly up to 1d upstream of the object, before it decreases due to the blockage effect. A mathematical correlation based on experimental data has been proposed to estimate the concentration of the diesel PM with respect to the wind velocity at different distances upstream of the objects.

[1]This study was supported with a grant from METRANS research program.

[2]Professor and Director

[3]Graduate Assistant

10:43AM H9.00002 Feature formation in the planar-flow melt spinning of metals linked to meniscus unpinning , ANTHONY ALTIERI, PAUL STEEN, Cornell University — In the planar-flow melt spinning of metals (PFMS), liquid metal is brought into contact with a cold, rotating wheel and thin ribbon is spun off. A unique characteristic of the PFMS process is the narrow gap region between the nozzle, where liquid metal is introduced, and the wheel, which allows for the formation of a liquid "puddle" held in place by surface tension. The flow behavior in the puddle can have a dramatic effect on the appearance of the thin ribbon product. In particular, thickness depressions which span the width of the ribbon appear periodically under certain casting conditions. These features are related to puddle vibrations of the air-liquid meniscus which forms upstream of the fluid inlet. A lower frequency (1 kHz) feature previously has been related to Rayleigh oscillations of a drop. We report that a higher frequency (10 kHz) feature, also dependent on Rayleigh oscillations, depends additionally on the translation speed of the substrate, as seen by correlation. This difference is related to meniscus pinning/unpinning and flow structures within the puddle. A model of fluid flow which connects these two features and a discussion of these differences will be presented.

10:56AM H9.00003 Design and characterization of an aerodynamic shoe sampling system for screening trace explosive materials , MATTHEW STAYMATES, National Institute of Standards and Technology, GREG GILLEN, JESSICA GRANDNER, STEFAN LUKOW — As part of an ongoing effort with the Transportation Security Laboratory, the National Institute of Standards and Technology has been developing a prototype shoe sampling system that relies on aerodynamic sampling as the primary mechanism for liberating, transporting, and collecting explosive contamination. This presentation will focus on the fluid dynamics associated with the current prototype design. This design includes several air jets and air blades that are used to dislodge particles from target areas of a shoe. A large blower then draws air and liberated particles into a collection device at several hundred liters per second. Experiments that utilize optical particle counting techniques have shown that the applied shear forces from these jets are capable of liberating particles efficiently from shoe surfaces. Results from real-world contamination testing also support the effectiveness of air jet impingement in this prototype. Many examples of flow visualization will be shown. The issues associated with air spillage, particle release efficiency, and particle transport will also be discussed.

11:09AM H9.00004 Numerical simulation of a cross flow Marine Hydrokinetic turbine.[1] , TAYLOR HALL, ALBERTO ALISEDA, Mechanical Engineering. University of Washington. — In the search for alternative sources of energy, the kinetic energy of water currents in oceans, rivers and estuaries is being explored as predictable and environmentally benign. We are investigating the flow past a cross flow turbine in which a helical blade under hydrodynamic forces turns around a shaft perpendicular to the free stream. This type of turbine, while very different from the classical horizontal axis turbine commonly used in the wind energy field, presents advantages for stacking in very narrow constricted channels where the water currents are consistently high and therefore turbine installation may be economically feasible. We use a model of a helical four-bladed turbine in cross flow to investigate the efficiency of the energy capture and the dynamics of the turbulent wake. Scale model experiments in a flume are used to validate the numerical results on a stationary configuration as an initial step towards creating an accurate numerical model of the turbine. The simulation of the rotating turbine provides a full perspective on the effect of angular position on flow detachment and vortex shedding from the blade, as well as on the fluctuations of the shaft torque produced (a problematic feature of this type of turbine). The results are analyzed in terms of hydrodynamic optimization of the blade and its structural loading.

[1]Supported by DOE through the Northwest National Marine Renewable Energy Center

11:22AM H9.00005 Investigation of Helical Cross-Flow Axis Hydrokinetic Turbines, Including Effects of Waves and Turbulence , PETER BACHANT, MARTIN WOSNIK, University of New Hampshire — A new test bed for hydrokinetic turbines was used to evaluate different cross-flow axis turbines, and investigate effects of waves and turbulence. Turbine thrust (drag) and mechanical power were measured in a tow tank with cross section 3.7 x 2.4m at speeds of 0.6-1.5 m/s for a Gorlov Helical Turbine (GHT) and a Lucid spherical helical turbine (LST). GHT performance was also measured in progressive waves of various periods, grid turbulence, and in a cylinder wake. Overall, the GHT performs with higher power and thrust coefficients than the LST. A 2nd law, or kinetic exergy efficiency, defined as the fraction of kinetic energy removed from the flow that is converted to usable shaft work, was measured. The distribution of energy into shaft work and turbulent kinetic energy in the wake can affect environmental transport processes and performance of turbines arrays. Progressive waves generally enhance performance of the GHT, but can lead to stall at higher tip speed ratios compared to the steady case. Grid turbulence delays dynamic stall and enables operation at lower tip speed ratios, while not decreasing maximum power coefficient. Performance in a cylinder wake is highly dependent on the cylinder's cross-stream location, ranging from benign to detrimental. The experimental observations provide insight into the physical principles of operation of cross-flow axis turbines.

11:35AM H9.00006 Applying Turbulence Models to Hydroturbine Flows: A Sensitivity Analysis Using the GAMM Francis Turbine[1] , BRYAN LEWIS, JOHN CIMBALA, ALEX WOUDEN, Pennsylvania State University — Turbulence models are generally developed to study common academic geometries, such as flat plates and channels. Creating quality computational grids for such geometries is trivial, and allows stringent requirements to be met for boundary layer grid refinement. However, engineering applications, such as flow through hydroturbines, require the analysis of complex, highly curved geometries. To produce body-fitted grids for such geometries, the mesh quality requirements must be relaxed. Relaxing these requirements, along with the complexity of rotating flows, forces turbulence models to be employed beyond their developed scope. This study explores the solution sensitivity to boundary layer grid quality for various turbulence models and boundary conditions currently implemented in OpenFOAM. The following models are resented: k-omega, k-omega SST, k-epsilon, realizable k-epsilon, and RNG k-epsilon. Standard wall functions, adaptive wall functions, and sub-grid integration are compared using various grid refinements. The chosen geometry is the GAMM Francis Turbine because experimental data and comparison computational results are available for this turbine.

[1]This research was supported by a grant from the DoE and a National Defense Science and Engineering Graduate Fellowship.

11:48AM H9.00007 Numerical Simulations of Marine Hydrokinetic (MHK) Turbines Using the Blade Element Momentum Theory[1] , TEYMOUR JAVAHERCHI, OSKAR THULIN, ALBERTO ALISEDA, University of Washington — Energy extraction from the available kinetic energy in tidal currents via Marine Hydrokinetic (MHK) turbines has recently attracted scientists' attention as a highly predictable source of renewable energy. The strongest tidal resources have a concentrated nature that require close turbine spacing in a farm of MHK turbines. This tight spacing, however, will lead to interaction of the downstream turbines with the turbulent wake generated by upstream turbines. This interaction can significantly reduce the power generated and possibly result in structural failure before the expected service life is completed. Development of a numerical methodology to study the turbine-wake interaction can provide a tool for optimization of turbine spacing to maximize the power generated in turbine arrays. In this work, we will present numerical simulations of the flow field in a farm of horizontal axis MHK turbines using the Blade Element Momentum Theory (BEMT). We compare the value of integral variables (i.e. efficiency, power, torque and etc.) calculated for each turbine in the farm for different arrangements with varying streamwise and lateral offsets between turbines. We find that BEMT provides accurate estimates of turbine efficiency under uniform flow conditions, but overpredicts the efficiency of downstream turbines when they are strongly affected by the wakes.

[1]Supported by DOE through the National Northwest Marine Renewable Energy Center.

12:01PM H9.00008 Injection pumps , WILLIAM SCHULTZ, BOSUK HAN, ERIC JOHNSEN, University of Michigan — Injection pumps are simple devices with no moving parts but with complex flow including free-shear and boundary layers combined with shocks and possible multiphase flow. In spite of this, they are susceptible to simple control volume analyses. Here, we examine the extent to which these analyses represent optimal performance for a steam injector. We develop performance measures for 2D RANS computations and experiments.

12:14PM H9.00009 Spray Characterization of Gas-to-Liquid Synthetic Jet Fuels[1] , KUMARAN KANNAIYAN, REZA SADR, Texas A&M University at Qatar — In the recent years, development of alternative jet fuels is gaining importance owing to the demand for cleaner combustion. In addition to having energy density that matches those of conventional fuels, alternate jet fuels need to possess vital qualities such as rapid atomization and vaporization, quick re-ignition at high altitude, less emission, and poses ease of handling. The fuel preparatory steps (atomization and vaporization) and mixing in a combustion chamber play a crucial role on the subsequent combustion and emission characteristics. Gas-to-Liquid (GTL) synthetic jet fuel obtained from Fischer-Tropsch synthesis has grabbed the global attention due to its cleaner combustion characteristics as a result of the absence of aromatics and sulphur. As a part of an on-going joint effort between Texas A&M at Qatar (TAMUQ), Rolls-Royce (UK), and German Aerospace Laboratory (DLR), a spray characterization experimental facility is set up at TAMUQ to study the spray characteristics of GTL fuel and highlights the influence of change in fuel composition on the spray characteristics. In this work, spray characteristics such as droplet size, velocity, and distribution of different GTL fuel blends is investigated and compared with the spray characteristics of conventional JetA1 fuel.

[1]Supported by Qatar Science and Technology Park, QSTP.

12:27PM H9.00010 Removal of resist film from wafer surface by steam-water mixture jet , TAKASHI MASHIKO, TOSHIYUKI SANADA, Shizuoka University, ITSUO NISHIYAMA, Daipla Wintes, HIDEO HORIBE, Kanazawa Institute of Technology — We have shown that the steam-water mixture jet, a two-fluid jet with its carrier gas being steam, exhibits high cleaning performance when sprayed onto a target. This is a promising technique which requires only simple apparatus and little or no chemicals, but the cleaning mechanism remains unknown. We have conducted a series of experiments to elucidate the mechanism and learn how to meet given industrial requirements (e.g., set parameters for desired detergency). In our recent experiment, we adopted a resist-coated silicon wafer as the target and evaluated the jet performance of resist removal from the wafer. The removal performance proved to be a decreasing function of the resist hardness and of the resist-wafer adhesivity, and an increasing function of the jet duration. These results suggest that the resist removal by the steam-water mixture jet mainly consists of physical processes (e.g., peel-off process), in contrast to the traditional resist-removal techniques utilizing chemical reactions.

Monday, November 21, 2011 10:30AM - 12:14PM –
Session H10 MHD II 313

10:30AM H10.00001 Numerical study of laboratory MRI experiment , CHRISTOPHE GISSINGER, JEREMY GOODMAN, Princeton University, HANTAO JI, Princeton University, PPPL — Theoretical studies of the MagnetoRotational Instability (MRI) generally rely on a local description, or computations between axially infinite (or periodic) cylinders. Since laboratory MRI experiments involve finite geometries, it is important to understand the effect of boundaries on the MRI. We investigate numerically the flow of a conducting fluid in a Taylor-Couette flow when an axial magnetic field is applied. To minimize Ekman recirculation due to vertical no-slip boundaries, two rotating rings are used in the vertical endcaps, approximating setup used in the Princeton MRI experiment. Our 3D global simulations show that, in presence of boundaries, the nature of the bifurcation, the saturation and the structure of axisymmetric MRI modes are significantly affected by the resultant recirculation. In addition, large scale non-axisymmetric modes are obtained when the applied field is sufficiently strong. We show that these modes are related to destabilization of a free shear layer created by the conjugate action of the applied field and the rotating rings. Finally, we compare our calculations in cylindrical and spherical geometries to recent experimental results obtained in the Maryland experiment and the Princeton MRI experiment.

10:43AM H10.00002 Turbulent Magnetized Spherical Couette Flow , MATTHEW ADAMS, DANIEL ZIMMERMAN, SANTIAGO TRIANA, DANIEL LATHROP, University of Maryland, College Park — We present experimental studies of the turbulent flow of a conducting fluid in a spherical shear flow in the presence of a magnetic field. Our experimental apparatus consists of an outer spherical shell concentric with an inner sphere, each of which can be rotated independently. The geometry of the experiment makes these studies applicable to geophysical and astrophysical bodies. Liquid sodium serves as the working fluid, filling the gap between the inner sphere and the shell. By applying an axial magnetic field of varying strength, the influence of the applied field on the fluid flow can be studied. Measurements of the magnetic field around the device are used to extract information about the global fluid flow. We also measure the torque required to drive the inner and outer spheres at their respective rotation rates. For a variety of inner and outer sphere rotation rates, we observe enhanced angular momentum transport as the applied field strength is increased, and we compare this with the observed magnetic field pattern and expected magnetorotational instabilities.

10:56AM H10.00003 Experimental study of the Lorentz forces on a small magnet generated by a liquid metal flow , CHRISTIANE HEINICKE, GAUTAM PULUGUNDLA, ANDRÉ THESS, Institute of Thermodynamics and Fluid Mechanics, Ilmenau University of Technology — Liquid metals are hot and aggressive and therefore inaccessible to conventional flow measurement techniques. However, the determination of local and global flow properties of metal melts and electrolytes is of great industrial interest. Lorentz force velocimetry is a contactless measurement technique, based on the use of magnets, that is able to bridge this gap. While recent studies showed the suitability of this technique for global flow quantities, this project aims at reaching local resolution inside turbulent liquid metal flows. We can show, how the Lorentz force generated by the magnetic field inside the liquid metal depends on the distance of the magnet to the duct, and more importantly, on the flow velocity. Numerical analyses have been performed with a good agreement with the experimental results.

11:09AM H10.00004 Instabilities of thin layers of conducting fluids produced by time dependent magnetic fields , JAVIER BURGUETE, Universidad de Navarra — We present the recent results of an experiment where thin layers of conducting fluids are forced by time-dependent magnetic fields perpendicular to their surface. We use as conducting fluid an In-Ga-Sn alloy, immersed in a 5% hydrocloric acid solution to prevent oxidation. The conducting layers have a circular shape, and are placed inside a set-up that produces the vertical magnetic field. Due to MHD effects, the competition between the Lorentz force and gravity triggers an instability of the free surface. The shape of this surface can adopt many different configurations, with a very rich dynamics, presenting azimuthal wave numbers between 3 and 8 for the explored parameters. The magnetic field evolves harmonically with a frequency up to 10Hz, small enough to not to observe skin depth effects and with a magnitude up to 0.1 T. Different resonant regions have been observed, for narrow windows of the forcing frequency. We have analysed the existence of thresholds for these instabilities, depending on the wave number and experimental parameters. These results are compared with others present in the literature.

11:22AM H10.00005 LES of transitional duct flows by the non-uniform Lorentz force, HIROMICHI KOBAYASHI, Keio University, HIROKI SHIONOYA, YOSHIHIRO OKUNO, Tokyo Institute of Technology — Large-eddy simulation (LES) of turbulent duct flows is carried out in the liquid metal MHD power generator, and the influence of the non-uniform Lorentz force caused by the non-uniform magnetic flux density on the turbulent flows is examined. As increasing the high magnetic flux density, the structures of Reynolds stress align along the orientation of the magnetic flux density, and those structures periodically flowed toward the downstream region like Karman vortex sheets are observed. The stronger magnetic flux density makes the laminarized flow turbulent again. That is confirmed by using spectrum analysis. It is found that a pair of eddy as the secondary flow for the non-MHD duct flow diminishes in the MHD flow. The non-uniform magnetic flux density in the streamwise direction produces the eddy currents that lead to the M-shaped velocity profiles in the plane parallel to the external magnetic field. The velocity profiles are modulated more strongly with the magnetic flux density.

11:35AM H10.00006 Point dipole as a magnetic obstacle in liquid metal duct flow[1], SASKIA TYMPEL, THOMAS BOECK, DMITRY KRASNOV, JÖRG SCHUMACHER, TU Ilmenau — Lorentz force velocimetry is a new contactless technique to measure the velocities of hot and agressive conductiong liquids. The measurement of the Lorentz force on the magnet is highly sensitive to the velocity profile that is influenced by the magnetic field. Thus the knowlegde of the flow transformation and the influence of an inhomogeneous local magnetic field on liquid metal flow is essential for obtaining velocity information from the measured forces. We consider liquid metal flow in a square duct with electrically insulating walls under the influence of a magnetic point dipole using three-dimensional direct numerical simulations with a finite-difference method. The dipole acts as a magnetic obstacle. A wide range of parameters affects the created wake. In this canonical setting, we study the modification of the flow for different Hartmann and Reynolds numbers. We observe a strong dependence of the magnetic obstacle effect and the corresponding Lorentz force on the orientation of the dipole as well as on its position.

[1]The authors acknowledge the support of the Deutsche Forschungsgemeinschaft.

11:48AM H10.00007 ABSTRACT WITHDRAWN —

12:01PM H10.00008 Numerical Study of the Interaction of Turbulent Liquid Metal Flow with an Inhomogeneous Magnetic Field[1], GAUTAM PULUGUNDLA, CHRISTIANE HEINICKE, CHRISTIAN KARCHER, Institute of Thermodynamics and Fluid Mechanics, Ilmenau University of Technology — In this work, we present the numerical analysis of a turbulent liquid metal flow in the inhomogeneous magnetic field of a permanent magnet. The study is motivated by Lorentz Force Velocimetry (LFV), a non-contact technique for flow rate measurement of conducting fluids using complex magnet systems producing non-uniform and localised magnetic fields. As a simplified case, we consider the flow of liquid metal in a straight square duct with electrically insulating walls. For this configuration, numerical simulations are performed by coupling the commercial finite volume solver FLUENT and finite element solver COMSOL Multiphysics. Parametric analyses are performed with different flow Reynolds numbers and magnet positions. Furthermore, the numerical results are validated with experimental studies performed in the liquid metal laboratory at Ilmenau University of Technology. The analyses provide good reference results for the numerical calibration of LFV.

[1]Financial support from Deutsche Forschungsgemeinschaft (grant GRK 1567/1).

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H11 Turbulence Modeling II 314

10:30AM H11.00001 A dynamic global-coefficient mixed subgrid-scale model for large eddy simulation of turbulent flows, SATBIR SINGH, DONGHYUN YOU, Carnegie Mellon University — The dynamic global coefficient model of You and Moin [Phys. Fluids **19**, 065110 (2007)] is modified by employing the mixed closure of Bardina [PhD Thesis, Stanford University (1983)] as the base model. The new dynamic global-coefficient mixed model explicitly calculates the modified Leonard stress while utilizes the global-coefficient eddy viscosity model for the cross stress and subgrid-scale Reynolds stress. The dynamic procedure of determining the model coefficient is based on the "global-equilibrium" between the subgrid-scale dissipation and the viscous dissipation. The present model does not require averaging or *ad-hoc* clipping of the model coefficient, which is a highly desirable feature for large-eddy simulations in complex configurations. The model has potential to alleviate the assumption of the alignment of the subgrid- scale stress and the resolved strain rate tensor, while improving the capability for the local characteristics of subgrid-scale turbulence. In a number of test simulations, the new subgrid-scale model showed good predictive capability and was found to provide good representation of local characteristics of subgrid-scale turbulence.

10:43AM H11.00002 A subgrid-scale similarity-like model for LES with improved subgrid-scale dissipation, B.W. ANDERSON, J.A. DOMARADZKI, University of Southern California — The scale-similarity model in LES is based on the intuitive assumption that the velocities associated with different scales in the flow give rise to turbulent stresses that have a similar character, leading to the attractively simple functional form of the model. However, it is well known that scale-similarity models suffer from an insufficient SGS dissipation which causes them to fail in actual LES, negating advantages of the functional simplicity. We have developed a model given by an expression reminiscent of the classic similarity model in its functional simplicity but which also provides the adequate SGS dissipation. The proposed model is constructed by applying an explicit filter to the resolved velocity and splitting it into components associated with the largest and smallest resolved scales. Terms describing specific interscale interactions are then constructed using the test-filtered velocity field and these terms are used to formulate an alternative expression for SGS stress that predicts correctly the global energy flux from the resolved to subgrid scales. Results for wall-bounded flow simulations performed with this model indicate that accurate predictions of mean and rms flow quantities as well as SGS dissipation and energy budget are possible.

10:56AM H11.00003 Scale-separation models for the larger eddies in turbulent flow[1], ROEL VERSTAPPEN, University of Groningen — Large-eddy simulation (LES) seeks to predict the dynamics of spatially filtered turbulent flows. The very essence of LES is that the LES-field contains only scales of size $\geq \delta$, where δ denotes the (user-chosen) length of the spatial filter. In the present approach we continue the work that was conducted during the 2010 CTR Summer Program by addressing the following two basic questions: (a) when does a LES-model stop the production of smaller scales of motion from continuing at the filter scale; and (b) when does it dissipate any disturbances having a length scale smaller than δ initially. In this way we find two scale separation conditions that ensure that all subfilter scales are dynamically insignificant. These conditions can be applied to any type of LES-model. In case of a mixed model, for instance, they imply that the eddy viscosity ν_t has to depend on the invariants $q = \frac{1}{2}\mathrm{tr}(S2)$ and $r = -\frac{1}{3}\mathrm{tr}(S3)$ of the (filtered) strain rate tensor S. The simplest model is then given by $\nu_t = \frac{1}{96}\,\delta2|r|/q$. This model is successfully tested for a turbulent channel flow (Re_τ=590 and 1,000).

[1]This work was partially supported by the Dutch Compute Challenge Programme of NWO/NCF

11:09AM H11.00004 Investigation of numerical, resolution, and subgrid-scale model effects on LES predictive quality of a variable-density jet , GREGORY RODEBAUGH, LESTER SU, Johns Hopkins University — Large-eddy simulation (LES) provides a methodology to simulate complex, unsteady flows that are too expensive for DNS, and for which the RANS formulation gives poor results. To maximize the practical utility of LES, computations should be performed at the coarsest resolution that still provides satisfactory predictions. To have an a priori estimate of the required resolution for a given flow configuration, a thorough understanding of the nonlinear error interactions present in LES is needed; therefore, we seek to elucidate the effects of different subgrid-scale (SGS) stress and scalar flux models, at a range of resolution levels, on both the mixing properties and turbulent statistics of the flow. To isolate these effects in a non-homogeneous turbulent shear flow, we investigate an isothermal, axisymmetric turbulent jet, with either fixed or variable density, discretized by several explicit finite difference schemes in cylindrical coordinates. The results show that dynamic eddy viscosity based models for the momentum equation display less sensitivity to grid resolution than dynamic mixed models while capturing the mean flow well and reproducing fluctuations with reasonable accuracy. The mixture fraction evolution, however, is influenced negligibly by the choice of SGS scalar model at course to medium grid resolutions due to the use of upwind operators for its convective terms.

11:22AM H11.00005 Turbulence modeling for finite volume LES of channel flow with unresolved wall layers , HENRY CHANG, ROBERT MOSER, University of Texas at Austin — An incompressible turbulent channel flow is simulated using a staggered grid finite volume LES. The grid is uniform with $\Delta y^+ = 50$ and is therefore unresolved near the wall. Optimal large eddy simulation models (OLES) are applied with results that are similar to many other LES models near the wall. For example, the rms streamwise velocity fluctuations are greatly over-predicted. To address this issue, the OLES model is reformulated based on a Reynolds decomposition of the velocity, with OLES models for the mean and fluctuating fluxes formulated according to their statistical structure. These models yield much better LES results, with only a slight over-prediction of streamwise velocity fluctuations. This talk will cover the details of the models, the LES results, and the implications for LES models of wall-bounded flows.

11:35AM H11.00006 Explicit filtering based LES of turbulent combustion , COLIN HEYE, The University of Texas at Austin, COLLEEN M. KAUL, The University of Texas at Austin; Center for Turbulence Research, Stanford University, VENKAT RAMAN, The University of Texas at Austin — Large eddy simulation (LES) of turbulent combustion relies on accurate characterization of small-scale scalar dissipation rate and scalar variance. Subfilter models for these quantities are based on filter-scale gradients. In practical LES computations, the implicit filtering approach inextricably links the filter width to the computational mesh. Since numerical errors are highest at scales comparable to the mesh size, numerical errors severely contaminate the evaluation of important subfilter quantities. Here, an explicit filtering approach is developed for conserved scalar-based combustion applications, which allows the mesh to be varied independently of the filter size. Using both homogeneous and jet-type flows, the practical issues in such implementations are examined. In particular, the need for non-invertible filtering techniques is discussed.

11:48AM H11.00007 Computational Complexity of Coherent Vortex and Adaptive Large Eddy Simulations of Three-Dimensional Homogeneous Turbulence at High Reynolds Numbers[1] , ALIREZA NEJADMALAYERI, ALEXEI VEZOLAINEN, OLEG V. VASILYEV, University of Colorado at Boulder — With the recent development of parallel adaptive wavelet collocation method, adaptive numerical simulations of high Reynolds number turbulent flows have become feasible. The integration of turbulence modeling of different fidelity with adaptive wavelet methods results in a hierarchical approach for modeling and simulating turbulent flows in which all or most energetic parts of coherent eddies are dynamically resolved on self-adaptive computational grids, while modeling the effect of the unresolved incoherent or less energetic modes. This talk is the first attempt to estimate how spatial modes of both Coherent Vortex Simulations (CVS) and Stochastic Coherent Adaptive Large Eddy Simulations (SCALES) scale with Reynolds number. The computational complexity studies for both CVS and SCALES of linearly forced homogeneous turbulence are performed at effective non-adaptive resolutions of 256^3, 512^3, 1024^3, and 2048^3 corresponding to approximate Re_λ of 70, 120, 190, 320. The details of the simulations are discussed and the results of compression achieved by CVS and SCALES as well as scalability studies of the parallel algorithm for the aforementioned Taylor micro-scale Reynolds numbers are presented.

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

12:01PM H11.00008 Measurements of the Budget of the Subgrid-Scale Stress in Convective Atmospheric Surface Layers[1] , KHUONG NGUYEN, Clemson University, STEVEN ONCLEY, THOMAS HORST, PETER SULLIVAN, National Center for Atmospheric Research, CHENNING TONG, Clemson University — Measurement data obtained in the atmospheric surface layer during the recent Advection Horizontal Array Turbulence Study (AHATS) field program are used to obtain the subgrid-scale (SGS) stress budget. The array technique for obtaining the SGS stress was extended to include pressure sensors to measure the fluctuating pressure, enabling separation of the resolvable- and subgrid-scale pressure. The budget terms are analyzed using two parameters, the surface layer stability parameter -z/L, and the ratio of the wavelength corresponding to the peak of the vertical velocity spectrum to the filter size. For very large filter sizes, the budget terms show trends with -z/L that are similar to those of the Reynolds stress budget terms. For small filter sizes, the trends are very different, sometimes the opposite. The measured budget terms generally balance to within 10 percent. The results have implications for using model transport equations for SGS stress.

[1]Supported by NSF

12:14PM H11.00009 Implicit LES of turbulent lid-driven cavity flows , J.M. MCDONOUGH, Unversity of Kentucky — The lid-driven cavity (LDC) problem has long provided a canonical test of CFD codes, first in 2D and now in 3D. At least in part due to its importance in CFD (simple geometry and boundary conditions), it has also received considerable attention from experimentalists. Here, we employ this problem to demonstrate behavior of implicit LES (ILES) performed on very coarse computational grids (as few as 1000 grid cells) at Reynolds numbers near transition to turbulence, and higher. Both "standard" ILES and "enhanced" (via synthetic-velocity models) ILES results will be compared with those from laboratory experiments and with highly-resolved DNS for the basic LDC problem consisting of a cubical cavity with only one wall in constant motion. Of particular interest will be the ability, or lack thereof, of ILES to predict transition to turbulence at the appropriate Reynolds number, and the extent to which turbulent flow calculations match results observed in experiments as numerical discretizations are refined. It will be shown that synthetic-velocity enhanced ILES provides improved predictions over standard versions even for very simple synthetic-velocity models.

12:27PM H11.00010 Velocity-Resolved LES (VR-LES) technique for simulating turbulent transport of high Schmidt number passive scalars , SIDDHARTHA VERMA, GUILLAUME BLANQUART, California Institute of Technology, P.K. YEUNG COLLABORATION — Accurate simulation of high Schmidt number scalar transport in turbulent flows is essential to studying pollutant dispersion, weather, and several oceanic phenomena. Batchelor's theory governs scalar transport in such flows, but requires further validation at high Schmidt and high Reynolds numbers. To this end, we use a new approach with the velocity field fully resolved, but the scalar field only partially resolved. The grid used is fine enough to resolve scales up to the viscous-convective subrange where the decaying slope of the scalar spectrum becomes constant. This places the cutoff wavenumber between the Kolmogorov scale and the Batchelor scale. The subgrid scale terms, which affect transport at the supergrid scales, are modeled under the assumption that velocity fluctuations are negligible beyond this cutoff wavenumber. To ascertain the validity of this technique, we performed a-priori testing on existing DNS data. This Velocity-Resolved LES (VR-LES) technique significantly reduces the computational cost of turbulent simulations of high Schmidt number scalars, and yet provides valuable information of the scalar spectrum in the viscous-convective subrange.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H12 Turbulence: Shear Layers I 315

10:30AM H12.00001 Statistics and scaling of turbulence in a spatially developing shear layer, ANTONIO ATTILI, FABRIZIO BISETTI, King Abdullah University of Science and Technology — A direct numerical simulation of a turbulent mixing layer has been performed. The flow achieves a Reynolds number based on Taylor's microscale equal to 250. The flow originates from a laminar, hyperbolic tangent inlet profile, which is perturbed with white noise. The turbulence statistics in the self-similar state agree with a number of previous results, where the flow evolves from turbulent inlets. Our data suggests that the final state of the layer could be universal. The degree of convergence to high Reynolds number asymptotic behavior has been evaluated by analyzing spectra and second order structure functions. The scaling exponents of high order structure functions are calculated using the Extended Self-Similarity. Over a wide range of scales, the exponents are found to match those in homogeneous isotropic turbulence. However, an additional scaling range is found at larger scales, where the effects of mean shear and coherent large structures are not negligible. The values of the exponents recovered in this second range agree well with those in flows with strong shear (boundary layers, channels and wakes).

10:43AM H12.00002 Generation of a buoyant jet by a sphere moving vertically in a stratified fluid, HIDESHI HANAZAKI, Kyoto University, KEISUKE SHIMOBATA, HIROYASU YOSHIKAWA — We consider the flow past a sphere moving vertically at a constant speed in salt-stratified fluid. Experiments have shown that vertical jets are generated even if the vertical mean density gradient is not large. At least seven types of wake structures have been found, including a thin jet with a surrounding bell-shaped structure. In this numerical study, we investigate the unsteady generation mechanism of buoyant jets with the focus on the diffusion effects of the density/salt. When a sphere descends in a stratified fluid, the density is initially conserved along the movement of the fluid and the originally horizontal isopycnal surfaces are simply deformed vertically and pulled down by the sphere. As time proceeds, diffusive effects become significant in the density boudary layer on the sphere surface and in the thin jet, and the density is no longer conserved. This violation of conservation would be the origin of the jet which is composed of the fluids which moves up to return to thier original heights.

10:56AM H12.00003 Turbulence and wave motion driven by horizontal shear layers below/above stably stratified regions, MOHAMED MOUSTAOOUI, Arizopna State University, JULIAN C.R. HUNT, University College London, ALEX MAHALOV, Arizona State University — Analytical studies and high resolution 3-dimensional numerical simulations of a jet show how eddies in turbulent shear flows moving with mean velocity ΔU relative to external upper or lower stably stratified regions with buoyancy frequency N_o interact with these regions. Internal waves are not generated if the Richardson number is less than a critical value *Ri**. If *Ri*< *Ri*$'$ the stratification only perturbs the irrotational motions outside the shear layer. But if *Ri*$'$ <*Ri*<*Ri** intense interfacial shear layers forms at the top of the shear layer and then a second layer forms at a height of order w^+/N_o, that bounds a mixed layer of trapped turbulent eddies and weak mean shear. This double layer structure has similar scales as the natural layering process in turbulent stratified flows. When *Ri*>*Ri**, first the upper and then the lower interfacial layers break up, the peak shear in the interface decreases and a narrow band spectrum of internal waves are generated which propagate upwards although they do not significantly interact with the turbulence in the shear layer. The Reynolds shear stress associated with the waves varies with *Ri*, but over a wide range of *Ri* is as much as about 1/5 of the peak shear in the shear layer; when normalised on the vertical variance at the interface it increases in proportion to$\sqrt{Ri-1}$.

11:09AM H12.00004 The effects of weak stratification on turbulence within a mixing layer, ROI GURKA, Ben-Gurion University, ELIEZER KIT, Tel-Aviv University — In mixing-layers between two parallel streams of different densities, shear and gravity effects interplay; buoyancy acts as a restoring force and the Kelvin Helmholtz mode is known to be stabilized by the stratification. The effect of stratification on streamwise and spanwise vortical structures in the stratified mixing layer has been investigated numerically (DNS) and experimentally (PIV). Two cases were compared: homogeneous flow (Ri=0) vs. weakly stratified (Ri=0.03). Mixing of the heated upper layer with unheated lower layer occurred in the wake of the splitter plate with attached oscillating Chevron type flappers. Controlling the attached to splitter plate flappers enabled to form different type of disturbances (2D and 3D) followed by various types of instabilities within the mixing layer. It was found that the energy of 3D modes at the initial stage increased faster in the case of stratified flow compared to the homogeneous case, indicating that some of the modes are convectively unstable and growth faster than they do in homogeneous flow. In addition, the lateral RMS distribution of the velocity components in stratified case was wider indicating faster spreading of the mixing layer. Those points out to convective instability within the primary roll caused by the density layers overturn occurring throughout the roll formation.

11:22AM H12.00005 Viscous and inviscid velocity contributions near the turbulent/nonturbulent interface in a planar turbulent jet, FILIPE SOARES PEREIRA, CARLOS DA SILVA, IST - Technical University of Lisbon, GERRIT ELSINGA, JERRY WESTERWEEL, TU - Delft — Direct numerical simulations (DNS) of turbulent planar jets and shear free turbulence are used to assess the viscous and inviscid contributions to the entrainment velocity at the turbulent/non-turbulent (T/NT) interface, and their dependency on the Reynolds number and flow type. Whereas in shear free turbulence the viscous contribution dominates in the turbulent planar jet the inviscid contribution is more important. Furthermore, it is shown that as the Reynolds number increases the inviscid contribution becomes larger in both flows. Moreover it was observed that the for the planar turbulent jet configuration the indirect estimation of the T/NT interface thickness using the local velocity does not match the T/NT interface thickness as observed from the conditional vorticity profile.

11:35AM H12.00006 Passive scalar dynamics near the turbulent/nonturbulent interface in a jet, RODRIGO R. TAVEIRA, CARLOS DA SILVA, IST - Technical University of Lisbon — The present work uses several direct numerical simulations (DNS) of turbulent planar jets at Reynolds number ranging from $Re_\lambda = 120$ to $Re_\lambda = 160$ and Schmidt numbers raging from $Sc = 0.7$ to 7.0 to analyze the nature and properties of the *"scalar interface"* and to investigate the dynamics of turbulent mixing of a passive scalar. Specifically, we employ conditional statistics in relation to the distance from the T/NT interface in order to eliminate the intermittency that affects common turbulence statistics close to the jet edge. The physical mechanisms behind scalar mixing near the T/NT interfaces and their associated turbulent scales and topology are investigated. A sharp scalar interface exists separating the Turbulent and the irrotational flow regions. The thickness of this scalar interface δ_θ is also of the order of the Taylor micro-scale, λ. However, the thickness of the scalar gradient variance $\left\langle \theta^2 \right\rangle_I$ (where $G_j = \partial\theta/\partial x_j$) is much smaller. Very intense scalar gradient sheet structures along regions of intense strain, in particular at the T/NT interface. The scalar gradient transport equation is analyzed in order to further investigate the physical mechanism of scalar turbulent mixing at the jet edge. Almost all mixing takes place in a confined region close to the interface, beyond which they become reduced to an almost in perfect - balance between production and dissipation of scalar variance.

11:48AM H12.00007 ABSTRACT WITHDRAWN —

12:01PM H12.00008 Passive Scalar Dispersion in Uniformly Sheared Turbulence , CHRISTINA VANDERWEL, STAVROS TAVOULARIS, University of Ottawa — The dispersion of a plume of dye from a continuous point source is investigated experimentally in uniformly sheared, nearly homogeneous, turbulent flow generated in a water tunnel. The flow has a turbulence Reynolds number of $Re_\lambda \approx 150$ and a shear rate parameter of $S^* \approx 12$. Neutrally buoyant fluorescent dye (fluorescein and/or rhodamine B/6G) is injected isokinetically from a streamlined injection tube into the developed turbulent flow. Instantaneous concentration and velocity fields of the plume are measured simultaneously using planar laser induced fluorescence (PLIF) and particle image velocimetry (PIV). Joint statistics of the concentration and velocity fields are measured and compared to predictions of theoretical models of both absolute and relative (two-particle) dispersion. The contribution to scalar transport by horseshoe vortices, shown earlier to be the dominant coherent structures of uniformly sheared turbulence, is determined by analysis of coincident instantaneous maps of concentration and velocity.

12:14PM H12.00009 Similarity Theory for an Axisymmetric Turbulent Wake with Rotation , MARTIN WOSNIK, University of New Hampshire — Axisymmetric wakes are special cases of turbulent shear flows in the sense that the local Reynolds number based on velocity deficit and wake width decreases with downstream position. Recently, Johansson et al. (Physics of Fluids, **15**, no.3, 603-617, 2003) showed that two distinct similarity solutions for the non-swirling axisymmetric turbulent wake exist – one for infinite and one for low local Reynolds number. Every axisymmetric wake, no matter how high the initial Reynolds number, will eventually transition to the low Reynolds number similarity state in the far wake. Here equilibrium similarity considerations are applied to axisymmetric turbulent wakes with rotation (swirl), as can be found downstream of wind or hydrokinetic turbines. By examining under which conditions the reduced momentum and Reynolds stress transport equations for swirling wakes as well as the momentum integrals admit to similarity solutions, asymptotic scaling relations for the decay of velocity deficit and swirl are found. Swirl is introduced as an initial condition, and additional constraints on the similarity solution are introduced from the turbine (wake generator) operating parameters, e.g., tip speed ratio, angular induction, etc. The consequences of having a non-point source of thrust (drag) and angular momentum are investigated. Implications of the findings on the operation of wind and hydrokinetic turbines and turbine arrays are discussed.

12:27PM H12.00010 Pressure time correlation in turbulent shear flows , LI GUO, LNM, Institute of Mechanics, Chinese Academy of Sciences, XIN ZHANG, GUOWEI HE, LNM, Institute of Mechanics, Chinese Academy of Science — Pressure time correlations are important to turbulence-generated noise and flow-induced vibration. In this study, we numerically and theoretically investigate pressure time correlations in turbulent shear flows. We calculate the pressure time correlation from the direct numerical simulation (DNS) of turbulent channel flows. It is observed that the pressure time correlations decay faster than the streamwise velocity time correlations. This is different from isotropic homogenous turbulence, where the pressures have the same decorrelation time scales as velocities. A theoretical model is developed to confirm this observation. We further check Taylor frozen-flow model for pressure time correlation in turbulent channel flows. It is found that Taylor's model is not valid for strong shear rates. Based on this observation, we develop a non-frozen flow model, which takes the mean shear effects into account. This model is also verified using the DNS data of turbulent channel flows.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H13 Multiphase Flow IV: Numerical I 316

10:30AM H13.00001 Numerical simulation of a round liquid jet using the refined level set grid method with subgrid Lagrangian drop breakup model[1] , DOKYUN KIM, PARVIZ MOIN, CTR, Stanford University — An accurate and robust numerical method has been developed to simulate the breakup of a round liquid jet surrounded by a coaxial flow of gas. A Refined Level Set Grid (RLSG) method coupled to a Lagrangian drop breakup model is used to capture the breakup process of the liquid jet. The phase interface is tracked by the level-set method, while small subgrid droplets produced from resolved ligaments are transferred from the level-set representation to the Lagrangian drops. The further secondary atomization is handled by a stochastic breakup model. When thin ligaments are not resolved on the level-set grid, a capillary breakup model is used to predict the drop size distribution from the pinching off process and inserted as Lagrangian drops. This method improves the mass conservation error as well as reducing the computational cost. The numerical results are consistent with the observed breakup mechanisms in the experiment and the stability analysis. The drop size distribution of the resulting spray is also compared with the experimental data. These numerical results demonstrate the applicability and feasibility of our method for simulation of the atomization process of liquid jets.

[1]Supported by the Office of Naval Research

10:43AM H13.00002 A multilevel simulation approach to derive the slip boundary condition of the solid phase in two-fluid models[1] , ZHI-GANG FENG, University of Texas at San Antonio, EFSTATHIOS MICHAELIDES, SHAOLIN MAO, Los Alamos National Laboratory — The simulation of particulate flows for industrial applications often requires the use of a two-fluid model (TFM), where the solid particles are considered as a separate continuous phase. One of the underlining uncertainties in the use of aTFM in multiphase computations comes from the boundary condition of the solid phase. The no-slip condition at a solid boundary is not a valid assumption for the solid phase. Instead, several researchers advocate a slip condition as a more appropriate boundary condition. However, the question on the selection of an exact slip length or a slip velocity coefficient is still unanswered. In the present work we propose a multilevel simulation approach to compute the slip length that is applicable to a TFM. We investigate the motion of a number of particles near a vertical solid wall, while the particles are in fluidization using a direct numerical simulation (DNS); the positions and velocities of the particles are being tracked and analyzed at each time step. It is found that the time- and vertical-space averaged values of the particle velocities converge, yielding velocity profiles that can be used to deduce the particle slip length close to a solid wall.

[1]This work was supported by a grant from the DOE-NETL (DE-NT0008064) and by a grant from NSF (HRD-0932339).

10:56AM H13.00003 ABSTRACT WITHDRAWN –

11:09AM H13.00004 Numerical investigation of a turbulent hydraulic jump: Interface statistics and air entrainment[1] , MILAD MORTAZAVI, Graduate student at Stanford University, DOKYUN KIM, Post-doctoral fellow at Stanford University, ALI MANI, Post-doctoral fellow at MIT, PARVIZ MOIN, Franklin and Caroline Johnson Professor of Mechanical Engineering, Stanford University — The objective of this study is to develop an understanding of formation of bubbles due to turbulence/interface interactions and nonlinear surface wave phenomena. As a model problem a statistically stationary turbulent hydraulic jump has been considered. Turbulent hydraulic jump with an inflow Froude number of 2 and Reynolds number of 88000–based on inflow height–has been numerically simulated. Based on typical air- water systems, a density ratio of 831 has been selected for our calculations. A refined level-set method is employed to track the detailed dynamics of the interface evolution. Comparison of flow statistics with experimental results of Murzyn et al. (Int. J. Multiphase Flow, 2005) will be presented. The probability density function of principal curvatures of the air- water interface and curvature distribution patterns in the chaotic regions are investigated. The importance of liquid impact events in bubble generation will be discussed.

[1]Supported by the Office of Naval Research, with Dr. Pat Purtell, program manager.

11:22AM H13.00005 Multiscale modeling of blood-plasma separation in bifurcations[1] , XUEJIN LI, Brown University, ALEKSANDER POPEL, Johns Hopkins University, GEORGE KARNIADAKIS, Brown University — Motion of a suspension of red blood cells (RBCs) flowing in a Y-shaped bifurcating microfluidic channel is investigated using a low-dimensional RBC validated 3D model based on dissipative particle dynamics. No-slip wall boundary and adaptive boundary conditions were implemented to model hydrodynamic flow within a specific wall structure of diverging microfluidic channels. Plasma skimming and the all-or-nothing phenomenon of RBCs in a bifurcating microfluidic channel have been investigated in our simulations, including the size of cell-free layer on the daughter channels. The results show that the flowrate ratio of the daughter channels and the feed hematocrit level have considerable influence on blood-plasma separation. Compared with the particle recovery efficiencies of healthy RBCs, malaria-infected RBCs (iRBCs) have a tendency to travel into the low flowrate daughter channels because of the increased stiffness of iRBCs. The simulation results are consistent with previous experimental results and theoretical predictions.

[1]This work was suporrted by the National Institute of Health (NIH), Grant R01Hl094270 and the National Science Foundation (NSF) Grant CBET-0852948.

11:35AM H13.00006 Direct numerical simulations of gas-liquid annular flows in horizontal pipes: predictions of film height and mechanisms for film sustainment[1] , JEREMY MCCASLIN, OLIVIER DESJARDINS, Cornell University — Direct Steam Generation (DSG), a technology that uses parabolic solar reflectors to generate steam from water flowing through horizontal pipes located at the focal points of the reflectors, often requires an annular pipe flow in which the liquid is distributed as a thin film around the circumference of the pipe. The distribution of the gas-liquid interface for such flows (i.e. the thickness of the liquid film and the measure of liquid droplets entrained in the gas core) can have ramifications for both the optimized operation and economical design of DSG loops. In this work, a conservative finite difference scheme is used in conjunction with a state-of-the-art discontinuous Galerkin conservative level set methodology to simulate periodic sections of such flows. Under the assumption of a gas core-dominated flow, dimensional analysis suggests a theoretical basis that is presented for the prediction of flow "annularity" (i.e. contiguousness of the liquid film). Mechanisms for film sustainment such as wave propagation up the pipe walls and droplet entrainment and deposition are also numerically investigated for a variety of annular flows.

[1]This research was supported in part by the NSF through TeraGrid resources provided by the National Institute for Computational Sciences.

11:48AM H13.00007 High order discretization of interfacial jump conditions for the Poisson equation on Cartesian grids , AMIR RIAZ, University of Maryland, KEEGAN DELANEY, ELIAS BALARAS, George Washington University — A robust, 2^{nd} order accurate discretization method for both Dirichlet and Neumann jump conditions at sharp discontinuities in space is developed for the Poisson equation with spatially varying, discontinuous coefficients. The method advances previous approaches that are based on either 1^{st} order treatment of jumps across discontinuities or employ implementations that are not robust for 2-D and 3-D applications with moving interfaces. This particularly simple and robust approach to higher order discretization is based on the volume-fraction weighted average of the solution variables at cell centers. The resulting coefficient matrix for the Poisson equation remains symmetric and can be inverted by the available fast solver algorithms. Examples and algorithmic details will be discussed.

12:01PM H13.00008 Numerical modeling of the early interaction of a planar shock with a dense particle field , JONATHAN REGELE, GUILLAUME BLANQUART, California Institute of Technology — Dense compressible multiphase flows are of interest for multiphase turbomachinary and energetic material detonations. Still, there is little understanding of the detailed interaction mechanisms between shock waves and dense (particle volume fraction $\alpha_d > 0.001$) particle fields. A recent experimental study [Wagner et al, AIAA Aero. Sci., Orlando, 2011-188] has focused on the impingement of a planar shock wave on a dense particle curtain. In the present work, numerical solutions of the Euler equations in one and two dimensions are performed for a planar shock wave impinging on a fixed particle curtain and are compared to the experimental data for early times. Comparison of the one- and two-dimensional results demonstrate that the one-dimensional description captures the large scale flow behavior, but is inadequate to capture all the details observed in the experiments. The two-dimensional solutions are shown to reproduce the experimentally observed flow structures and provide insight into how these details originate.

12:14PM H13.00009 High Fidelity Simulation of Liquid Jet in Cross-flow Using High Performance Computing , MARIOS SOTERIOU, XIAOYI LI, United Technologies Research Center — High fidelity, first principles simulation of atomization of a liquid jet by a fast cross-flowing gas can help reveal the controlling physics of this complicated two-phase flow of engineering interest. The turn-around execution time of such a simulation is prohibitively long using typically available computational resources today (i.e. parallel systems with ~O(100) CPUs). This is due to multiscale nature of the problem which requires the use of fine grids and time steps. In this work we present results from such a simulation performed on a state of the art massively parallel system available at Oakridge Leadership Computing Facility (OLCF). Scalability of the computational algorithm to ~2000 CPUs is demonstrated on grids of up to 200 million nodes. As a result, a simulation at intermediate Weber number becomes possible on this system. Results are in agreement with detailed experiment measurements of liquid column trajectory, breakup location, surface wavelength, onset of surface stripping as well as droplet size and velocity after primary breakup. Moreover, this uniform grid simulation is used as a base case for further code enhancement by evaluating the feasibility of employing Adaptive Mesh Refinement (AMR) near the liquid-gas interface as a means of mitigating computational cost.

12:27PM H13.00010 Towards direct numerical simulation of pressure swirl injectors with realistic geometries , MARK CZAJKOWSKI, OLIVIER DESJARDINS, Cornell University — Atomization of hydrocarbon fuels is of critical importance to the transportation sector, in particular for aircraft gas turbine engines. In this work, simulations of a Delevan pressure swirl injector with realistic geometry was investigated. Results were compared with simulations performed by Fuster et al. (Int J Multiphase Flow, 2009) of a swirl jet at lower density ratios. The pressure swirl injector is used for many applications and is a component within air-blast injectors commonly found in gas turbines and aeroengines. Direct numerical simulation of the pressure swirl injection process has the potential to provide much-needed information about the complex physics of atomization in swirling flows, but has yet to be used due to the interaction of a complex turbulent multiphase flow with complicated injector geometries. A variety of novel numerical methods are used to facilitate the numerical simulations including a conservative implementation of immersed boundaries used to represent the injector geometry, an accurate interface transport scheme with mass conservation properties based on a discontinuous Galerkin discretization of the conservative level set method, and a novel discretization of the Navier-Stokes convective term allowing for robust simulations at high density ratios. Simulations were conducted by combining the methods with a fully parallelized computational code called NGA.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H14 Bubbles II 317

10:30AM H14.00001 Two Fates in Bubble Breakup against Azimuthal Perturbations

, LIPENG LAI, WENDY W. ZHANG, James Franck Institute & Physics Department, The University of Chicago — Cylindrically-symmetric breakup of an air bubble underwater is unstable against azimuthal perturbations. As perturbations in the initial shape of the bubble neck excite vibrations that persist over time, the breakup dynamics always becomes nonlinear. As a consequence, experiments find various breakups with small variation in control parameters. Here we analyze a simplified process where the dynamics is perturbed by an n=2 single mode and the only variation is the initial phase of perturbation. Using boundary integral method, our simulation classifies qualitatively different breakups into two categories. One is characterized by a topological change of the bubble neck via smooth contact between different parts of air-water interface. This dynamics is dominated by lower fundamental vibrational modes. The other is characterized by the formation of near cusp (high local curvature) along the interface which requires contribution from higher modes. Switching from one outcome (smooth contact) to the other (highly curved surface) as initial phase varies coincides with transition from constructive to destructive interference between wave modes 2 and 4.

10:43AM H14.00002 Description of the instability transition modes of a bubble rising in still liquids[1]

, J.C. CANO-LOZANO, P. BOHORQUEZ, C. MARTÍNEZ-BAZÁN, Universidad de Jaen — In this research, we have investigated numerically the transition from straight to zigzag motion during the rise of a single gas bubble of diameter D in a pure-clear stagnant liquid, for the limiting case $\rho_g/\rho_l \ll 1$ and $\mu_g/\mu_l \ll 1$, where ρ is density, μ is dynamic viscosity, and subindices g and l denote gas and liquid phases, respectively. The transition is determined in terms of the Reynolds, $Re = \rho_l\, g^{1/2}\, D^{3/2}/\mu_l$, and Bond, $Bo = \rho_l\, g\, D^2/\sigma$, numbers as set of nondimensional, independent parameters governing the flow dynamics, in which g is the acceleration due to gravity and σ is the surface tension. Subsequently, the neutral curve for the onset of zigzag motion is characterized in the $\{Re, Bo\}$-plane.The transition curve is determined computing the real shape of the ascending bubble. Thus, we discuss the effect of the bubble shape and aspect ratio on the instability characteristics of the bubble wake. In particular, we observed that the onset of the zigzag instability begins with the loss of axisymmetry of the wake, developing two counter-rotating vortices, which exhibit a symmetry plane.

[1]Supported by the Spanish MICINN, Junta de Andalucía and EU Funds under projects DPI2008-06624-C03-02 and P07-TEP02693.

10:56AM H14.00003 Numerical Simulation of Bubble Dynamics in Deformable Vessels[1]

, VEDRAN CORALIC, TIM COLONIUS, California Institute of Technology — The growth and collapse of cavitation bubbles has been implicated as a potential damage mechanism leading to the rupture of blood vessels in shock wave lithotripsy (SWL). While this phenomenon has been investigated numerically, the resulting simulations have often assumed some degree of symmetry and have often failed to include a large number of influential physics, such as viscosity, compressibility, surface tension, phase change and fluid-structure interactions. We present here our efforts to explore the role that cavitation bubbles play in the rupture of blood vessels in SWL and to improve upon the current state of the numerical approach. We have developed a three- dimensional, high-order accurate, shock- and interface- capturing, multicomponent flow algorithm that accounts for the effects of viscosity and surface tension. At this time, we omit any effects due to elasticity and instead, as a first step, model tissue as a viscous and stiffened gas. We discuss preliminary results for the Rayleigh and shock-induced collapse of a gas bubble within a blood vessel and characterize the increase in vessel deformation with increasing bubble confinement and proximity to the vessel wall.

[1]This research was supported by the National Institutes of Health grant No. 2PO1DK43881.

11:09AM H14.00004 Toroidal Bubble Rings Interactions in Viscous Fluids

, JING LOU, MING CHENG, Institute of High Performance Computing, T.T. LIM, National University of Singapore — We investigate rising toroidal bubbles in viscous fluids by using a 3-D Lattice Boltzmann model simulation, as well as experimental measurement. The bubble ring behavior is systematically characterized with effects of bubble vortex strength, buoyancy force, interface tension and viscosity dissipation. The bubble rise velocity, bubble spread/diffusion and instability is also modeled and compared with experimental observations. Further on, we modeled two co-centered rising toroidal bubbles interaction in a transient manner. In particular, the complex vortex interaction and its impact on bubble pair and flow wakes are discussed.

11:22AM H14.00005 Dynamics of air bubbles passing through a liquid-liquid interface

, ROMAIN BONHOMME, Institut de Radioprotection et de Surete Nucleaire, JACQUES MAGNAUDET, Institut de Mecanique des Fluides de Toulouse, BRUNO PIAR, Institut de Radioprotection et de Surete Nucleaire, INTERFACE TEAM, DPAM/SEMIC/LIMSI TEAM — The passage of rising air bubbles through an initially flat horizontal liquid-liquid interface is studied using both laboratory experiments and Direct Numerical Simulation. The dynamics of spheroidal, spherical cap and toroidal bubbles near the liquid-liquid interface and subsequently through the upper liquid are investigated by coupling high-speed shadowgraph visualizations and Particle Image Velocimetry techniques. Axisymmetric computations are also carried out to assess the validity of presently available computational approaches in three-phase flows. These computations are based on two distinct approaches, namely a Volume Of Fluid approach without interface reconstruction and a Cahn-Hilliard model coupled with the incompressible Navier-Stokes equations. Experimental and computational results are compared in various configurations, including cases where the bubble is trapped at the liquid-liquid interface or rises in the upper phase while towing a column of the lower liquid that eventually breaks into droplets of various sizes.

11:35AM H14.00006 Bubble Oscillations under Forced Vibrations

, MOHAMMAD MOVASSAT, NASSER ASHGRIZ, MARKUS BUSSMANN, University of Toronto — Dynamics of a gas bubble in a liquid container in response to forced vibrations is studied. A 3D two-fluid solver is employed to study the bubble behavior. Forced vibration induces an oscillatory buoyancy force on the bubble. In response to the forcing, bubble undergoes both oscillatory translational motion as well as shape deformation. As the amplitude and frequency of oscillations increase, the bubble response goes from a regular and linear behavior to a chaotic and nonlinear region in which large deformations occur and different shape modes are excited. As the forcing increases, the inertia force, due to the momentum of the surrounding liquid, starts to form a liquid jet within the bubble core. Surface tension force may not be strong enough to prevent the formation and penetration of the jet and the liquid jet may pierce the bubble forming a toroidal bubble shape with a liquid core in the middle.

11:48AM H14.00007 Thermal Lattice Boltzmann Simulations for Vapor-Liquid Two-Phase Flows in Two Dimensions[1]

, YIKUN WEI, YUEHONG QIAN, Institute of Applied Mathematics and Mechanics, Shanghai University, Shanghai, China, 200072 — A lattice Boltzmann model with double distribution functions is developed to simulate thermal vapor-liquid two-phase flows. In this model, the so-called mesoscopic inter-particle pseudo-potential for the single component multi-phase lattice Boltzmann model is used to simulate the fluid dynamics and the internal energy field is simulated by using a energy distribution function. Theoretical results for large-scale dynamics including the internal energy equation can be derived and numerical results for the coexistence curve of vapor-liquid systems are in good agreement with the theoretical predictions. It is shown from numerical simulations that the model has the ability to mimic phase transitions, bubbly flows and slugging flows.

[1]This research is support in part by the grant of Education Ministry of China IRT0844 and the grant of Shanghai CST 11XD1402300.

12:01PM H14.00008 An asymptotically consistent diffuse interface method for simulating bubble dynamics in inhomogeneous environments , A. TIWARI, C. PANTANO, J.B. FREUND, University of Illinois at Urbana–Champaign — Theoretical models are effective for describing the physics of spherically symmetric bubble collapse, rebound and oscillation. However, for multi-bubble problems (bubble clouds) or near a solid wall, linear Bjerknes theory can only explain weak bubble interactions that do not lead to significant deviations from spherical bubble. Full-domain-based numerical methods are, in principle, capable of describing the nonlinear dynamics of bubble interactions in these inhomogeneous environments, but simulation of even a single bubble collapse poses significant computational challenges resulting from the inherent dynamical instability of such bubbles. We present an efficient and geometrically flexible numerical method to study bubble interactions. The method is build around a five-equation Eulerian diffuse-interface flow model, asymptotically reduced from Baer–Nunziato's compressible multi-fluid model within the 3D finite-volume framework. A consistent compression technique is developed to prevent the smearing of volume fraction and density across the interface. Upon being rigorously tested for spherically symmetric bubble collapses, we use it to simulate near-wall collapses. We show the distinctive formation of a torus bubble as the involuting jet from the distal surface penetrates the proximal surface. Preliminary results for multi-bubble interactions are also presented.

12:14PM H14.00009 Numerical Study on Focusing of Ultrasounds in Microbubble-enhanced HIFU , YOICHIRO MATSUMOTO, KOHEI OKITA, SHU TAKAGI, The University of Tokyo — The injection of microbubbles into the target tissue enhances tissue heating in High-Intensity Focused Ultrasound therapy, via inertial cavitation. The control of the inertial cavitation is required to achieve the efficient tissue ablation. Microbubbles between a transducer and a target disturb the ultrasound propagation depending on the conditions. A method to clear such microbubbles has been proposed by Kajiyama et al. [Physics Procedia 3 (2010) 305-314]. In the method, the irradiation of intense ultrasounds with a burst waveform fragmentize microbubbles in the pathways before the irradiation of ultrasounds for tissue heating. The vitro experiment using a gel containing microbubbles has showed that the method enables to heat the target correctly by controlling the microbubble distribution. Following the experiment, we simulate the focusing of ultrasounds through a mixture containing microbubbles with considering the size and number density distributions in space. The numerical simulation shows that the movement of the heating region from the transducer side to the target by controlling the microbubble distributions. The numerical results elucidate well the experimental ones.

12:27PM H14.00010 A train of rising Bretherton bubbles[1] , MICHAEL J. DAVIS, PETER S. STEWART, STEPHEN H. DAVIS, Northwestern University — Abstract: A closely fitting gas bubble in a vertically aligned capillary tube will rise due to buoyancy when the Bond number exceeds a critical value in the limit of low Capillary number, as shown by Bretherton [*J. Fluid Mech.* **10**, 1961]. We consider a steadily propagating train of such bubbles at various separation distances, and examine the additional influence of a temperature gradient along the walls of the tube. This problem is applicable to processes for manufacturing porous metal solids, where molten foams with low liquid fraction are solidified by an applied temperature drop. We seek to show that gravity can act as a means of control of the porosity of the foam as the liquid is cooled to its melting point.

[1]Work supported by NSF RTG grant DMS-0636574

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H15 Free-Surface Flows II 318

10:30AM H15.00001 A surface PIV approach for the remote monitoring of mean and turbulent flow properties in an open channel , ERIKA JOHNSON, EDWIN COWEN, Cornell University — In an effort to develop a reliable, continuous and efficient method of remotely monitoring bathymetry, water column turbulence levels and mean velocities, a surface PIV (particle image velocimetry) experiment is conducted in a wide open channel ($B/h > 12$) for a range of flow depths. Mean velocity, turbulence intensities, dissipation, divergence, vorticity as well as integral length scale statistics have been calculated from the surface PIV data. The results reveal the presence of secondary flow within the channel, which leads to heterogeneous turbulence metrics on the surface; for example, the streamwise turbulent intensities and dissipation vary strongly as a result of the secondary motion. The integral length scales vary predictably with the flow depth ($L \approx 0.3h$) and a correlation between the surface dissipation and dissipation in the water column allows estimates of the bed shear stress. These findings have important implications for developing new technologies for stream gauging, near shore and estuarine monitoring.

10:43AM H15.00002 Statistical study of scalar transport in wind-driven free-surface turbulent flows , HAMID REZA KHAKPOUR, LIAN SHEN, TAKERU IGUSA, Johns Hopkins University — We examine statistical methods for the study of the correlation between characteristic turbulence structures and scalar transport process in free-surface turbulence. We perform direct numerical simulation for the advection and diffusion of passive scalars in wind-driven free-surface turbulent shear flows. Using conditional averaging of events with strong scalar surface flux or large vorticity components, we characterize the correlation of surface flux with a variety of subsurface vortical structures. We then present a clustering method based on the Expectation-Maximization (EM) Algorithm. This clustering method is found to be effective in identifying dominant flow patterns which are associated with characteristic vortical structures. The method also provides insights into the induction of upwelling by vortical structures which, in turn, greatly enhances the scalar surface flux. The clustering method is general and can also be used for applications beyond interfacial scalar transport.

10:56AM H15.00003 The thermal and hydrodynamic structure of a turbulent buoyant jet on clean and contaminated free-surfaces , K. PETER JUDD, IVAN SAVELYEV, GEOFFREY SMITH, NRL — The thermal and hydrodynamic structure of a turbulent buoyant jet impinging normal to clean and contaminated free-surfaces was examined experimentally for fixed jet depth, reduced gravity and several Reynolds numbers. The objective of this investigation is to describe the resulting interaction and morphology of the surface thermal structures. Fluid for the jet is supplied from a gravity feed whose ambient temperature is several degrees above the receiving fluid of a large water basin. Thus the warmer fluid serves as a passive marker. The spatial and temporal characteristics of the surface thermal field were mapped using a mid-wave infrared imager sensitive to radiation in the 3-5 micron band and with an NEDT of 25 mK. As the Reynolds number and/or the degree of contamination are changed, noticeable structural changes were observed in the thermal field around the core and the outer turbulent/non- turbulent regions. Additionally, the subsurface jet was simultaneously interrogated using DPIV and the surface thermal structures are discussed in light of the resulting characteristics of the flow field.

11:09AM H15.00004 Large Eddy Simulation of Free-Surface Flow past a Submerged Submarine Fairwater at Moderate Reynolds Number , ZAHEER IKRAM, ELDAD AVITAL, JOHN WILLIAMS, Queen Mary University of London — The affects of reducing submergence depth around a submerged submarine fairwater without its associated appendages is numerically studied using Large Eddy Simulation. The submerged body is modelled using a immersed boundary method, whilst the free-surface is accounted for using a moving mesh. The numerical simulations are performed at a Reynolds number of 8×10^4 for a submergence ratio in the range of 0.44-0.32 and for Froude numbers < 1. Both the statistical and structural behaviour of the flow are examined. Statistically, time averaged velocity profiles, turbulence intensities, kinetic energy spectra and budgets have shown that the major part of the turbulence is confined to the near wake region of the fairwater. Structurally, no vortices are found to show significant rise or interaction with the free-surface, whilst in the wake vortices are found to be present for over 50% of the total monitored period. Reducing the submergence depth is found to influence the tip vortex. In all cases, the surface waves generated by the submerged fairwater are of a Kelvin kind.

11:22AM H15.00005 Harnessing sloshing as a passive dampener[1] , TAYLOR KILLIAN, ROBERT KLAUS, TADD TRUSCOTT, Brigham Young University — This study investigates the impact dynamics of hollow elastic spheres partially filled with fluid, similar to roller hockey balls. Unlike an empty elastic ball, the fluid mitigates some of the rebound through an impulse driven exchange of energy wherein the fluid is forced into a jet inside the ball. Images gathered through experimentation show that the fluid reacts more quickly to the impact than the ball, which decouples the two masses (fluid and ball), imparts energy to the fluid, and removes rebound energy from the ball. The experimental results are compared to an energy method where energy is transferred from the external motion of the ball, to the internal flow of the fluid. Results suggest that while the internal liquid affects the fluid motion, the rebound characteristics of the ball are uniform for a given amount of fluid. Implications of this work on an analog to the roller hockey ball is a potential use of similar passive dampening systems in sports technology and marine engineering.

[1]BYU ORCA

11:35AM H15.00006 Experimental investigation of turbulent wall jet , MATTHIEU A. ANDRE, PHILIPPE M. BARDET — Water jet flowing on a flat plate surrounded by quiescent air constitutes a standard case for the study of the interaction between turbulence and the liquid-air interface. This is of particular interest in the understanding of heat and mass transfers across interfaces. The structure of the surface has a great influence on the rate of the transfers which is critical for chemical processes like separation or absorption; pool-type nuclear reactor; climate modeling etc. This study focuses on high Froude (8 to 12) and Weber (3300 to 7400) numbers at which the surface exhibits small wavelength and large amplitude deformations, such as ligaments, surface break up with air entrainment and droplets projection. The experiment features a high velocity (up to 7.5 m/s) water wall jet (19.05mm thick at the nozzle exit) flowing on a flat plate ($Re = 10^5$ to $1.5 \cdot 10^5$). High speed movies and PLIF visualization show the evolution of the surface from smooth to 2D structures, then 3D disturbances as the turbulence arising from the wall interacts with the surface.

11:48AM H15.00007 Free surface shape and flows of an inviscid liquid , LAUST TOPHØJ, TOMAS BOHR, Technical University of Denmark, FLUID DTU TEAM — When the Euler equations governing inviscid flow are projected onto a stationary free surface, one can obtain a self-contained set of equations involving only the values of the physical fields on the free surface. These can be solved to yield relations between the shape of the free surface and the surface-tangential flow components at the free surface. Applications of a simplified set for systems with circular symmetry include [Bergmann et. al., JFM 679, pp. 415-431 (2011)]. The talk will focus on the derivation of these surface flow equations, with the free surface treated as a Riemann manifold. Examples of applications will be discussed.

12:01PM H15.00008 "Oenodynamic": hydrodynamic of wine swirling[1] , MARTINO RECLARI, MATTHIEU DREYER, STEPHANIE TISSOT, DANAIL OBRESCHKOW, FLORIAN WURM, MOHAMED FARHAT, Ecole Polytechnique Federale de Lausanne — A crucial step in wine tasting is the so called "swirling," necessary to release the bouquet of the wine: a gentle circular movement of the glass generates a wave propagating along the glass walls, enhancing oxygenation and mixing. Although being used in a large variety of other applications (e.g. cells cultures in orbital shaken bioreactors) this motion is not yet well understood. Using a simplified model we experimentally investigated the shape of the free surface and the mixing, and we identified a group of dimensionless parameters governing the flow.

[1]SNSF Grant CRSII2_125444

12:14PM H15.00009 Standing ripple rings within the super-critical flow region of a circular hydraulic jump , XIYU DU, PAUL STANLEY, Beloit College — The circular hydraulic jump in the presence of a soluble surfactant can display a stable, narrow ring of static ripple features within the inner super-critical flow region. These ripples are likely capillary surface waves pushed inward from the region of the hydraulic jump by surface tension gradient Marangoni type stresses. The behavior appears to be similar to that of the fluid pipe observed when a stream of pure water strikes a reservoir contaminated by a surfactant. In this experiment we primarily consider soluble surfactant dissolved in the source for the impinging jet, so that jet and downstream fluid have same surfactant concentration. We examine the effects of the surfactant concentration on the behavior of the jump and the inner static ripple feature, and propose a model as to the cause and location of these static ripples.

12:27PM H15.00010 Shallow flow down a spiral channel of small torsion and rectangular cross section[1] , YVONNE STOKES, HAYDEN TRONNOLONE, The University of Adelaide, STEPHEN WILSON, BRIAN DUFFY, University of Strathclyde Glasgow — Motivated by a desire to understand better the operation of spiral gravity separators, we consider flow of small depth down a helically wound channel of small torsion and rectangular cross section. The small fluid depth makes experimental investigation difficult, so that mathematical modelling and theoretical and computational studies are of great value for determining how such flows are influenced by fluid properties and geometrical parameters. We present a thin-film model, comprised of a system of non-linear ordinary differential equations, and show that there is a limiting value of a dimensionless parameter, defined in terms of fluid properties and geometrical parameters, beyond which there is no physically meaningful solution. Thin-film model solutions are compared with numerical solutions of a PDE model that does not assume small fluid depth.

[1]Funding from the UK EPSRC (Research Grant GR/S71873) and the University of Adelaide Special Studies Program to support visits by YMS to the University of Strathclyde is gratefully acknowledged.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H16 Porous Media II 319

10:30AM H16.00001 Reaction-driven viscous fingering , A. DE WIT, Nonlinear Physical Chemistry Unit, Universite Libre de Bruxelles, Brussels, Belgium (ULB), L.A. RIOLFO, ULB, S. IWATA, Nagoya Institute of Technology, 466-8555, Japan (NIT), R. MAES, P.M.J. TREVELYAN, ULB, Y. NAGATSU, NIT, ULB TEAM, NIT TEAM — An experimental demonstration of reaction-driven viscous fingering developing when a more viscous solution of a reactant A displaces a less viscous miscible solution of another reactant B is presented. In the absence of reaction, such a displacement of one fluid by another more mobile one is classically stable. However, a simple $A + B \rightarrow C$ reaction can destabilize this interface if the product C is either more or less viscous than both reactant solutions. Using the pH dependence of the viscosity of some polymer solutions, we provide experimental evidence of both scenarios. We demonstrate quantitatively that reactive viscous fingering results from the build-up in time of non-monotonic viscosity profiles with patterns behind or ahead of the reaction zone respectively depending on whether the product is more or less viscous than the reactants. The experimental findings are backed up by numerical simulations.

10:43AM H16.00002 Overlimiting current and deionization shocks in porous media[1] , MARTIN Z. BAZANT, DAOSHENG S. DENG, ALI MANI, E. VICTORIA DYDEK, Department of Chemical Engineering, MIT — Salt transport in bulk electrolytes occurs by diffusion and convection, but in microfluidic devices and porous media, surface conduction and electro-osmotic flow also contribute to ionic fluxes. The classical theory of electrokinetic phenomena in porous media assumes linear response to a small voltage, where the electrolyte concentration is only weakly perturbed. When a large voltage or concentration gradient is imposed, some surprising nonlinear electrokinetic phenomena result from the competition between bulk and interfacial transport in a microstructure. At constant voltage, the microstructure can sustain an over-limiting current (exceeding diffusion limitation) without any hydrodynamic or chemical instability. At constant current, a "deionization shock" can propagate through the microstructure, leaving behind a macroscopic region depleted of ions and particles. This talk will present experimental evidence for surface-driven overlimiting current and deioniziation shocks in porous glass frits, interpreted with the help of mathematical models, and applications to water deionization by "shock electrodialysis."

[1]Supported by the MIT Energy Initiative.

10:56AM H16.00003 Autocatalytic Reaction and Flow in Porous Media , SEVERINE ATIS, HAROLD AURADOU, LAURENT TALON, CNRS, DOMINIQUE SALIN, UPMC — Universités Pierre et Marie Curie, Paris Sud and CNRS. Laboratoire FAST, Bâtiment 502, UPS, 91405 ORSAY Cedex France. Coupling between autocatalytic reaction front and simple hydrodynamic flows leads to front patterns revealing the underlying flow field [1]. Flow of a passive tracer, i. e. a dye, through the complex flow field of a porous medium leads to the so-called hydrodynamic dispersion which accounts for the mixing process inside the medium [2]. We have performed experiments and numerical simulations of the propagation of reaction front in a porous medium. We have analyzed the dependences of the shape and velocity of the stationary fronts with the flow rate for a flow either in the same direction than the chemical front propagation or opposite to it. We determine the structure and characteristics of the front as well as the velocity distribution measured along the front. As a result this active, chemical, tracer allows to access to the characteristic of the complex flow field of the porous medium.

[1] M. Leconte, J. Martin, N. Rakotomalala and D. Salin. Pattern of reaction diffusion front in laminar flow. Phys. Rev. Letter., 90, 128302 (2003).
[2] J. C. Bacri, N. Rakotomalala and D. Salin. Experimental evidence of disorder effects in hydrodynamic dispersion. Phys. Rev. Lett. 58, 2035 (1987).

11:09AM H16.00004 Hydrodynamic instability induced by a precipitation reaction in a Hele-Shaw cell , YUKI ISHII, YUICHIRO NAGATSU, Nagoya Institute of Technology, 466-8555, Japan, ANNE DE WIT, Nonlinear Physical Chemistry Unit, CP 231, Université Libre de Bruxelles (ULB), 1050 Brussels, Belgium — We experimentally demonstrate hydrodynamic fingering destabilization of the interface between two miscible solutions of a reactant A and another reactant B respectively and of same viscosity upon displacement in a Hele-Shaw cell. The instability is driven by an A+B→C precipitation reaction producing a solid C and thus changing the local value of the permeability in the cell. The fingering pattern are observed to be different depending on whether A displaces B or vice-versa. A reaction-diffusion-convection (RDC) model describing the related dynamics is proposed. The origin of the instability is explained on the basis of the underlying mobility profile depending on the one-dimensional reaction-diffusion concentration profiles. Nonlinear simulations of the related RDC model reproduce the experimentally observed instability and explain that the asymmetric characteristics of the patterns depending on whether A displaces B or vice-versa are due to differences in diffusivity properties of A and B.

11:22AM H16.00005 ABSTRACT WITHDRAWN —

11:35AM H16.00006 Source-like Solution for Radial Imbibition into a Homogeneous Semi-Infinite Porous Medium , DANIEL ATTINGER, Department of Mechanical Engineering, Iowa State University, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University, JUNFENG XIAO, Department of Mechanical Engineering, Columbia University — We describe the imbibition process from a point source into a homogeneous semi-infinite porous material. When body forces are negligible, the advance of the wetting front is driven by capillary pressure and resisted by viscous forces. Our analytical results show that the absorbed volume flow rate is approximately constant with respect to time, and that the wetting front assumes a hemispherical shape with radius evolving in time as $r \sim t^{1/3}$. This cube-root law is confirmed by experiments using a packed cell of glass microspheres with average diameter of 42 μm. This result complements the one-dimensional imbibition law known as the Lucas-Washburn law where the imbibition length l evolves as $t^{1/2}$, and studies in axisymmetric porous cone with small opening angle by Reyssat et al. [1] where $l \sim t^{1/4}$ at long times.

[1] Reyssat, M., L. Courbin, E. Reyssat, and H.A. Stone, *Imbibition in geometries with axial variations*. J. Fluid Mech., 2008. 615: p. 335-344.

11:48AM H16.00007 Characterization and Performance of the Electroosmotic Pumping Effect for Different Porous Media , DANIEL PIWOWAR, THOMAS HANSEN, FRANCISCO DIEZ, Rutgers University — This work evaluates the electroosmotic pumping effect of various types of porous media. The focus was to determine the highest flow rate of each porous media and to find the maximum flow rate per surface area. The characterizations included power performance, flow rate, and back pressure measurements. Pumps are assembled in-house and included glass frits with varying thicknesses, glass fiber filters, porous anodic alumina membranes (PAAMS), and microcapillary arrays. Flow rates as high as 200mL/min are obtained. The flowrates for all porous media are normalized by the electric field and by the surface area to show the most efficient electroosmotic pumps. Results are in good agreement with recent published work. Closed loop pump systems are compared to open loop pump systems showing significant differences in flow rate and performance.

12:01PM H16.00008 Gas driven displacement in a Hele-Shaw cell with chemical reaction , ANDREW WHITE, THOMAS WARD, North Carolina State University — Injecting a less viscous fluid into a more viscous fluid produces instabilities in the form of fingering which grow radially from the less viscous injection point (Saffman & Taylor, Proc. R. Soc. Lon. A, 1958). For two non-reacting fluids in a radial Hele-Shaw cell the ability of the gas phase to penetrate the liquid phase is largely dependent on the gap height, liquid viscosity and gas pressure. In contrast combining two reactive fluids such as aqueous calcium hydroxide and carbon dioxide, which form a precipitate, presents a more complex but technically relevant system. As the two species react calcium carbonate precipitates and increases the aqueous phase visocosity. This change in viscosity may have a significant impact on how the gas phase penetrates the liquid phase. Experimental are performed in a radial Hele-Shaw cell with gap heights O(10-100) microns by loading a single drop of aqueous calcium hydroxide and injecting carbon dioxide into the drop. The calcium hydroxide concentration, carbon dioxide pressure and gap height are varied and images of the gas penetration are analyzed to determine residual film thickness and bursting times.

12:14PM H16.00009 Effects of CO_2 Solubility on Density and Mineral Trapping in Saline Aquifers , MOHAMMAD ALIZADEH NOMELI, AMIR RIAZ, University of Maryland College Park — We seek to characterize the thermodynamic behavior of CO_2 when stored in saline aquifers and predict how much CO_2 will be stored as mineral formations after a specific period of time. For this purpose, the PVTx model is used to simulate how temperature, pressure and salinity affect the solubility and mineral trapping of CO_2. Increasing temperature and salinity tend to lower the solubility of CO_2. It is also found that at low temperatures, the density of the ternary H_2O-CO_2-NaCl solution does not vary monotonically with pressure but displays a minimum which is proportional to salinity. At high temperatures, on the other hand, density increases monotonically with pressure and is inversely proportional to salinity. We present a model to find the dissolution and precipitation rates of minerals by taking into account the pressure, temperature, salinity and pH of the system. The model is validated with experimental data available from the literature.

12:27PM H16.00010 Optimal contant time injection policy for enhanced oil recovery and characterization of optimal viscous profiles[1] , PRABIR DARIPA, Texas A&M University — We numerically investigate the optimal viscous profile in constant time injection policy of enhanced oil recovery. In particular, we investigate the effect of a combination of interfacial and layer instabilities in three-layer porous media flow on the overall growth of instabilities and thereby characterize the optimal viscous profile. Results based on monotonic and non-monotonic viscous profiles will be presented. Time permitting. we will also present results on multi-layer porous media flows for Newtonian and non-Newtonian fluids and compare the results.

[1]The support of Qatar National Fund under a QNRF Grant is acknowledged.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H17 Geophysical Flows: General IV 320

10:30AM H17.00001 Large-eddy simulation of high Reynolds number gravity and turbidity currents , SENTHIL RADHAKRISHNAN, MOHAMAD NASR-AZADANI, ECKART MEIBURG, UC Santa Barbara — Gravity and turbidity currents are compositional and particulate laden flows that originate due to the horizontal pressure gradient. High Reynolds number currents are expensive to compute using Direct Numerical Simulation (DNS). Large-eddy simulation (LES) that uses sub-grid parameterization and approximate wall boundary condition provide a cheaper alternative. We have implemented the dynamic Smagorinsky model for sub-grid parameterization and a wall-layer model based on the law of the wall. These parameterizations are employed to simulate gravity and turbidity currents in lock-exchange configuration at high Reynolds number representative of laboratory and field scale currents. We consider current propagation over a flat surface and also past a complex topography. Results from these simulations, in particular, the front evolution and deposit profiles for various values of settling velocity will be discussed.

10:43AM H17.00002 Zonal jets in thermodynamic shallow water equations with radiative relaxation[1] , PAUL DELLAR, EMMA WARNEFORD, University of Oxford — The thermodynamic shallow water equations treat the reduced gravity as an advected scalar. This allows them to support horizontal variations in temperature, as well as the usual variations in depth and horizontal velocity. Originally proposed for lakes and tropical oceans, we have used them to model the atmospheres of gas giant planets. Adding a radiative cooling term to the temperature evolution equation creates a mass- and momentum-conserving shallow water model that produces super-rotating equatorial jets in our numerical simulations, unlike the sub-rotating jets found in standard shallow water simulations on rotating spheres. We also derive a quasigeostropic version of these equations, in which radiative coupling substantially increases the fraction of kinetic energy accounted for by the zonally-averaged flow in forced-dissipative simulations. Finally, we derive further reduced models in the limit of rapid radiative relaxation.

[1]Partially supported by EPSRC ARF grant EP/E054625/1

10:56AM H17.00003 Physical balances in non-hydrostatic balanced quasi-geostrophic equations , KEITH JULIEN, ANTONIO RUBIO, University of Colorado Boulder, IAN GROOMS, New York University — A suite of high resolution DNS for Rotating Rayleigh Benard convection using the non-hydrostatic balanced geostophic equations (NHBGE) (Julien et al 1998, 2006) over a wide range of parameter space was conducted to allow for the analysis of the term-by-term balances of the prognostic equations. These reduced equations offer greater access to the Low Rossby limit of thermal convection in comparison to the DNS of incompressible Navier-Stokes. The result has been a greater understanding of the transition between three regimes (cellular, columns, geostrophic turbulence) in terms of physical balances. Particularly, analysis reveals a dynamically unstable thermal boundary layer that can be modeled and turbulent interiors for which scalings with thermal forcing can be understood. Further insight has been gained into the scaling and breakdown of the convective Taylor columns (CTCs) (Grooms et al PRL, 2011) and the equation balances inside and outside of the CTCs in regimes where they coexist with cellular solutions.

11:09AM H17.00004 The Investigation of Wind Waves in Persian Gulf by Multi-level Long-term Hindcasts and In-situ Measurements , YING-PO LIAO, Zachry Department of Civil Engineering, Texas A&M University, College Station TX, USA, ARINDAM SINGHA, Department of Mechanical Engineering, Texas A&M University, Education City, Doha, Qatar, JAMES M. KAIHATU, Zachry Department of Civil Engineering, Texas A&M University, College Station TX, USA, REZA SADR, Department of Mechanical Engineering, Texas A&M University, Education City, Doha, Qatar — In this study, we employ both numerical and experimental methodologies to investigate the spatially and seasonally varying wind-wave features in the Persian Gulf and near Qatar due to the seasonal shamal. We perform a multi-level nested long-term wave hindcast using the SWAN wave model and five years of wind data from COAMPS (2004-2008), with emphasis on the area near Doha Port. A weather station, a stack of anemometers, and two video cameras are installed at Doha Port and will measure wind velocity profiles and wave kinematics/dynamics. These will be used to validate the climatology, as well as providing new insights into the physics of wind wave generation.

11:22AM H17.00005 Walking with coffee: when and why coffee spills , HANS C. MAYER, ROUSLAN KRECHETNIKOV, University of California at Santa Barbara — In our busy lives, almost all of us have to walk with a cup of coffee. Needless to say, under certain conditions we spill that precious liquid. This is a common example of the interplay between the mechanics of the complex motion of a walking individual and the fluid dynamics of a low viscosity liquid contained in a cup. We report on the results of an experimental investigation undertaken to explore the particular conditions under which coffee spills. Frame-by-frame analysis of recorded movies helps to elucidate the trajectory of the cup for various walking speeds and initial liquid levels. These kinematics, including both regular and irregular motions, are connected to instances during walking that result in spilled liquid. The coupling between mechanical aspects of walking and the fluid motion are analyzed based on which we determine a basic operational space with which one can confidently walk with cup in hand.

11:35AM H17.00006 Propagation of meteor infrasound in layered atmospheres , CHRISTOPHE MILLET, CHRISTOPHE P. HAYNES, CEA — In the present work, we provide a theoretical model of the shock wave generated by the atmospheric entry of meteors. Such a shock wave propagates from a strong blast wave region out to the far-field acoustic limit. The matching of the two regions is possible through a shape factor K, which is considered to be a random variable. The amplitude and period of the N-wave signal are obtained using Whitham's nonlinearization method for cylindrical waves. The method has been used to re-examine the crater-forming meteorite fall near Carancas, Peru (2007). As the specific trajectory of the meteor is unknown, all outcomes of the signal have been statistically analyzed. This includes finding the probability of a given signal and how the factor K affects this probability. Even though there is a good agreement between the period of the N-wave signal and the recorded signal, it is shown that a homogeneous isothermal atmospheric assumption leads to large errors in the ballistic overpressure relative to the observed values. Despite this, we demonstrate that inclusion of atmospheric absorption and meteoritic ablation within the model leads to the correct ground acoustic impulse. Future work involves a similar study for randomly layered atmospheres and will focus on the role of small-scale inhomogeneities.

11:48AM H17.00007 Lubrication Effect of Sea Spray under High-wind Conditions , YEVGENII RASTIGEJEV, N.C. A&T State University, SERGEY SUSLOV, Swinburne University of Technology, YUH-LANG LIN, N.C. A&T State University — Accurate modeling of sea spray under high-wind conditions is essential for improving intensity forecast of hurricanes or severe storms. It has been shown that the presence of water droplets in the vortex of hurricane leads to a significant reduction in turbulent intensity and consequently to a sharp flow acceleration. In this study, several mathematical models are proposed to detail the influence of sea spray on vertical momentum transport. The models are based on: turbulent kinetic energy (TKE) equation, E-ε closure and the Monin-Obukhov similarity (MOS) theory. It is demonstrated that for values larger than the critical speed, the spray concentration rapidly increases, which results in significant flow acceleration. All models produce qualitatively similar results for all turbulent flow parameters considered. It was found that the MOS-based and E-ε models tend to predict noticeably stronger lubrication effect than TKE model, especially for slower wind speeds. The results of calculations are in very good agreement with available experimental data. The work is supported by NOAA, NA06OAR4810187 and NSF, HRD-1036563 grants.

12:01PM H17.00008 Evolution of a Mushy Zone on a Finite Domain , NICHOLAS GEWECKE, University of Delaware, TIM SCHULZE, University of Tennessee — Mushy zones are regions of intermixed liquid and solid which result from instability due to the build-up of solute during solidification of multispecies materials. Transient dynamics in the case of a very cold lower boundary for a finite-depth tank lead to variations in the amount of solute which is frozen into the solid, which contrasts with the dynamics on an infinite domain. Furthermore, the growth of the solid layer eventually leads to the elimination of the mushy layer over very long time scales, which may be of interest in geological settings such as solidification of sea ice or magma chambers.

12:14PM H17.00009 Lava flow dynamics driven by temperature-dependent viscosity variations , SERINA DINIEGA, Jet Propulsion Laboratory, California Institute of Technology, SUZANNE SMREKAR, Jet Propulsion Laboratory, STEVEN ANDERSON, University of Northern Colorado, ELLEN STOFAN, Proxemy Research — As lava viscosity can change 1-2 orders of magnitude due to small changes in temperature, several studies have predicted the formation of low-viscosity/high-speed "fingers" (similar to a Saffman-Taylor type instability). Through the use of numerical simulation and steady-state analysis of model equations, we identify solutions that provide pahoehoe lava flows with a natural mechanism for the formation of channels within a sheet flow. We assume Hele-shaw-type geometry, Newtonian/laminar fluid flow, a Nahme's exponential law relating temperature and viscosity, and radiative heat-loss. Preliminary results indicate that flow-focusing occurs rapidly, but that the system settles into a new steady-state and does not create perpetually-lengthening hot-fingers. This suggests that additional physical processes are needed for the continued growth of preferred flow zones. This work has application to both Earth and planetary volcanology studies as the emplacement mechanics that yield long lava flows are not yet well understood.

12:27PM H17.00010 Mixing and entrainment in mantle plumes: A 3D experimental investigation , WILLIAM NEWSOME, ALINE COTEL, University of Michigan, CAROLINA LITHGOW-BERTELLONI, University College London, STANLEY HART, JOHN WHITEHEAD, Woods Hole Oceanographic Institution — Significant differences exist between isotopic signatures of typical mid-ocean ridge basalts (MORB) and those associated with many ocean islands, with ocean island basalts (OIB) generally exhibiting more variability in trace element concentrations and also a bias towards enrichment in radiogenic isotopes such as Sr, Nd, Hf and Pb. Such observations coupled with other geophysical evidence have been used to suggest that OIB's are surface manifestations of thermal plumes originating in the deep interior near the core-mantle boundary that interact with distinct, heterogeneous reservoirs as material is transported from the Earth's interior to the surface. We experimentally investigate the structure and transport characteristics of isolated thermal plumes in corn syrup. The 3D velocity field is measured using a scanning stereoscopic particle image velocimetry system. Two types of tracer particles are simultaneously utilized, with thermochromic liquid crystals providing an estimate of the temperature field. Lagrangian coherent structures computed from the velocity field identify key material lines and surfaces that provide a taxonomic picture of plumes operating in different regimes. These govern how the plume interacts with the ambient during its ascent.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H18 Microfluids: General V 321

10:30AM H18.00001 Hydrodynamic Coupling of Small Clusters of Particles in a Narrow Channel , WILLIAM USPAL, MATTHEW HELGESON, PATRICK DOYLE, Massachusetts Institute of Technology — Control of flowing suspensions is central to many emerging microfluidic applications. For instance, manipulation of small clusters is important in the synthesis of functional particles. Via theory, simulations, and experiment, we study small clusters confined in a microchannel with thin cross section and subject to an external flow. Our stop flow lithography (SFL) technique uniquely allows for precise control over particle shape, rigidity, and initial placement. We show that many-body hydrodynamic interactions sustain long-lived bound states with complex dynamics. As these interactions are sensitive to confinement, we investigate modulation of channel geometry as a means to perform sequential operations in a continuous process. We also probe the effects of shape and rigidity via SFL and a Lattice Boltzmann/Lattice Spring code, finding that near-field and orientational effects of shape enrich behavior. For soft, extended particles, we probe hydrodynamic self-deformation and self-excitation of elastic modes. Our results demonstrate phenomena that could be exploited for assembly of soft colloids in microchannels.

10:43AM H18.00002 Analysis of pinching in deterministic particle separation[1] , SUMEDH RISBUD, MINGXIANG LUO, JOELLE FRECHETTE, GERMAN DRAZER, Johns Hopkins University — We investigate the problem of spherical particles vertically settling parallel to Y-axis (under gravity), through a pinching gap created by an obstacle (spherical or cylindrical, center at the origin) and a wall (normal to X axis), to uncover the physics governing microfluidic separation techniques such as deterministic lateral displacement and pinched flow fractionation: (1) theoretically, by linearly superimposing the resistances offered by the wall and the obstacle separately, (2) computationally, using the lattice Boltzmann method for particulate systems and (3) experimentally, by conducting macroscopic experiments. Both, theory and simulations, show that for a given initial separation between the particle centre and the Y-axis, presence of a wall pushes the particles closer to the obstacle, than its absence. Experimentally, this is expected to result in an early onset of the short-range repulsive forces caused by solid-solid contact. We indeed observe such an early onset, which we quantify by measuring the asymmetry in the trajectories of the spherical particles around the obstacle.

[1]This work is partially supported by the National Science Foundation Grant Nos. CBET- 0731032, CMMI-0748094, and CBET-0954840.

10:56AM H18.00003 Manipulation of suspended microparticles by steady streaming , KWITAE CHONG, JEFF D. ELDREDGE, University of California, Los Angeles — It is well known that a body oscillating in a viscous fluid will generate steady streaming cells, a weak secondary flow created by non-linear interactions of the primary oscillatory flow. In this work, we explore the manner in which this steady streaming can transport microscale inertial particles. We use high-fidelity numerical simulations solving Navier-Stokes equation to simulate the steady streaming generated by a cylindrical probe undergoing translational oscillations perpendicular to its axis in a viscous fluid. It is observed that inertial particles are attracted toward small-amplitude limit cycles in one of the four after traveling a spiral trajectory. We especially focus on the influence of physical parameters such as inertial particle's size and density and probe oscillation frequency and amplitude on the streaming structures and attracting speed of the inertial particle. We also study the behaviors of particle motion in the vicinity of multiple probes, particularly when the oscillation parameters of the probes are asymmetric.

11:09AM H18.00004 Oscillations of a fiber flowing in a confined microchannel , HELENE BERTHET, ESPCI-PMMH, Schlumberger, MARC FERMIGIER, ESPCI-PMMH, GERARD DACCORD, Schlumberger, ANKE LINDNER, ESPCI-PMMH — Transport of slender bodies in confined geometries is of interest in various industrial applications. In the oil industry, fibers are widely used for stimulation or to prevent losses into the rock formations. Applications can also be found in biology systems such as targeted drug delivery. We present an experimental and numerical investigation of the flow of an advected fiber in a confined microchannel. The fiber is fabricated *in situ* using a photo-polymerization method to ensure an excellent control of its geometry and its mechanical properties. When imposing a constant flowrate, we observe that the fiber oscillates continuously between the lateral walls until it exits the channel. We characterize the oscillation period as a function of the flow velocity, the fiber length and channel width. This phenomenon can be used to generate efficient mixing at the microscale.

11:22AM H18.00005 Determining the equilibrium distribution of particles in nanofluidic systems , DAVID BOY, TODD SQUIRES, FREDERIC GIBOU, UC Santa Barbara — We study the equilibrium distribution of particles between plates with separation of less than one micron, which is relevant to many colloidal and biomolecular systems. We compare two common simplifying approximations, the linear superposition model and the ion model, to the full nonlinear Poisson-Boltzmann equation. We identify regions in which these simplified models apply, and we describe when and why they fail. Finally, we show that fitting a single parameter in a simplified model will cause it to agree with the full model, implying that even a believable fit is not evidence for a model's validity.

11:35AM H18.00006 Theory of viscous corrections to the acoustic radiation force on a suspended microparticle in a standing ultrasound wave , HENRIK BRUUS, MIKKEL SETTNES, Technical University of Denmark — We present a theoretical analysis of the acoustic radiation force causing acoustophoresis on suspended microparticles and cells in an standing ultrasound field of frequency ω. We include the kinematic viscosity ν of the solvent thereby extending the now classical and widely used theory by Gorkov valid only for inviscid solvents [1]. The viscosity appears through the formation of the incompressible viscous boundary layer of width a few times $\delta = \sqrt{2\nu/\omega}$ around the suspended particle. Previous analyses [2,3] of the dependence of δ had emphasis on developing general theoretical schemes and provided analytical expressions only in the limit $\delta \ll a \ll \lambda$. Our analysis does not have this limitation, and we take into account the incompressible boundary layer surrounding the particle and where viscosity dominates, and match the acoustic wave here with that in the compressible solvent where viscosity can be neglected. We apply our analytical result to calculate the values of the viscous corrections for particles of size and composition typically employed in microchannel acoustophoresis.

[1] L.P. Gorkov, Sov. Phys. Doklady **6**, 773 (1962).
[2] A.A. Doinikov. J. Acoust. Soc. Am. **101**(2), 722 (1997).
[3] S.D. Danilov, M.A. Mironov. J. Acoust. Soc. Am. **107**(1), 722 (2000).

11:48AM H18.00007 Particle-size dependent cross-over from radiation-dominated to streaming-dominated acoustophoresis in microchannels , RUNE BARNKOB, HENRIK BRUUS, Technical University of Denmark, PER AUGUSTSSON, THOMAS LAURELL, Lund University — Expanding the use of microchannel acoustophoresis to handle particles smaller than 1 μm is a challenge due to the particle-size dependent cross-over from the well-understood radiation-dominated motion of microparticles to the ill-characterized streaming-dominated motion of submicron particles. Using our newly-developed, automated, temperature-controlled, and high-precision micro-PIV system (Augusstson *et al.*, Lab Chip, submitted 2011), we measure the acoustophoretic velocity fields of polystyrene particles in the range from 10 μm to below 1 μm. We use the Helmholtz decomposition theorem and discrete Fourier transform to decompose each velocity field into a gradient part from radiation and a rotation part from streaming. From the decomposed velocity fields, we obtain as expected that the particle velocity induced by acoustic radiation scales with the particle size to the power 2, while the acoustic-streaming-induced particle velocity is independent of the particle size (Barnkob *et al.*, Proc. 14th MicroTAS 2010, p. 1247). Furthermore, we study the theoretical prediction that the critical particle size for cross-over scales linearly with the width of the viscous boundary layer and thus scales linearly with buffer viscosity and inversely with actuation frequency.

12:01PM H18.00008 Microfluidic modeling of the effects of nanoparticles on the blood-brain barrier in flow[1] , CRAIG SCHWAIT, University of Pennsylvania, RYAN HARTMAN, YUPING BAO, YAOLIN XU, University of Alabama — The difficulty of diffusing drugs across the blood-brain barrier (BBB) has caused an impasse for many brain treatments. Nanoparticles (NPs), to which drugs can adsorb, attach, or be entrapped, have the potential to deliver drugs past the BBB. Before nanoparticles can be used, their effects on the BBB and brain must be ascertained. Previous steady-state studies fall short for closely modeling *in vivo* conditions. Convection of nanoparticles is ignored, and endothelial cells' (ECs) morphology differs based on loading conditions; *in vitro* loading with continuous flow exhibit ECs indicating a more similar *in vivo* phenotype. NPs interact with monocytes prior to the BBB, and their toxicity effects were measured in flow conditions using both Trypan Blue cell counting and cell proliferation assays. The microfluidic device designed to model the BBB contained a concentric PES hollow fiber porous membrane in PFA tubing. Full use of the device will include ECs adhered on the inner surface and astrocytes adhered to the outer surface of the PES membrane to model cerebrovascular capillaries.

[1]Funded by NSF REU Site 1062611

12:14PM H18.00009 Diamagnetic Particle Deflection in Ferrofluid Flows through a Rectangular Microchannel , LITAO LIANG, XIANGCHUN XUAN, Clemson University — Magnetic field-induced particle manipulation is a promising technique for biomicrofluidics applications and offers several advantages over other traditional approaches based on electric, acoustic and optical forces. We present in this talk a fundamental study of diamagnetic particle motion in ferrofluid flows through a rectangular microchannel with a nearby permanent magnet. Due to their negligible magnetization relative to the ferrofluid, diamagnetic particles experience negative magnetophoresis and are repelled away from the magnet. The result is a three-dimensionally focused particle stream flowing near the bottom outer corner of the microchannel. The effects of particle size and position, ferrofluid flow rate and concentration, and magnet-channel distance on the diamagnetic particle deflection are systematically studied. The obtained experimental results agree reasonably with the predictions of a developed three-dimensional analytical model.

12:27PM H18.00010 ABSTRACT WITHDRAWN —

Monday, November 21, 2011 10:30AM - 12:40PM —
Session H19 Flow Control III 322

10:30AM H19.00001 Adjoint sensitivity analysis of time averaged quantities for unsteady flows[1]

, QIQI WANG, MIT — Sensitivity analysis is an essential gradient for data assimilation, aerodynamic design, uncertainty quantification and optimal flow control. In particular. the adjoint sensitivity analysis method has been shown to solve very high dimensional optimization problems typically found these applications. This talk focuses on recent developments in extending adjoint sensitivity analysis to unsteady flows. The adjoint equation of unsteady flows must be integrated backwards in time. Each backward time step must use the flow solution at the corresponding time. As a result, the entire time history of the flow solution must be either stored or recalculated. The invention of checkpointing schemes provides an economic solution to this challenge. In particular, the dynamic checkpointing scheme makes this solution more practical for computational fluid dynamics problems. In unsteady flows, the quantities of interest are often long time averages. We demonstrate that sensitivity analysis of these long time averaged quantities poses significant new challenge. A novel windowing scheme is developed to compute correct sensitivity for periodic unsteady flows, such as in laminar vortex shedding. Initial investigation of sensitivity analysis of chaotic unsteady flows, i.e., transitional and turbulent flows, is also discussed.

[1]This work is supported by a subcontract of PSAAP program at Stanford awarded to MIT.

10:43AM H19.00002 Optimal perturbations in plane Poiseuille flow based on the optimization of the p-norm of the energy density

, D.P.G. FOURES, C.P. CAULFIELD, DAMTP - University of Cambridge, P.J. SCHMID, LADHYX - Ecole Polytechnique — Over the last twenty years, much attention has been given to the consideration of transient non-modal growth of infinitesimal perturbations in shear flows. Indeed, it is now well-accepted that marked transient growth of the energy is possible for intermediate times even though all normal modes are linearly stable. It is often postulated that such amplification of an initial perturbation could trigger nonlinear behaviour within the flow and hence be responsible for the transition toward a turbulent state. Most of these studies have been based on the optimization of an initial perturbation in order to maximize the "gain," i.e. the amplification of the integrated kinetic energy over the flow domain. In many realistic circumstances, it is of more interest to identify initial perturbations which maximize flow quantities locally. Therefore, we investigate the implications of switching from the 1-norm of the energy density to a general p-norm which will approximate the infinity norm for large values of p. By considering a simple model problem of two-dimensional plane Poiseuille flow, we show that identifying "optimal" perturbations which maximize the p-norm of the energy density at some finite time horizon typically leads to enhanced localization, at the cost of reduction in the gain in the energy of the perturbation.

10:56AM H19.00003 ABSTRACT WITHDRAWN —

11:09AM H19.00004 Model reduction using snapshot-based realizations

, DIRK M. LUCHTENBURG, CLARENCE W. ROWLEY, Princeton University — A number of methods can be used to develop reduced-order models (ROM) for flow control, including proper orthogonal decomposition (POD), approximate snapshot-based balanced truncation (balanced POD), and the eigenvalue realization algorithm (ERA). We present a new method for obtaining a ROM from snapshots of the impulse response of a linear system. The idea is to use snapshots to identify a dynamical system that exactly reconstructs the impulse response. The dimension of this dynamical system is given by the rank of the controllable subspace (the number of states that respond to inputs). This system can be reduced even further by projecting out the unobservable subspace (the subspace that cannot be inferred from the output history), leading to a minimal realization. If the dimension of this realization is sufficiently small, balanced truncation can be readily performed. This is usually the case for active flow control applications, where the dimension of the flow field is large compared to the number of controllable / observable states. One advantage of the proposed method is that it avoids the construction of (large) Hankel matrices, as is necessary for methods like ERA or BPOD. Moreover, in general fewer snapshots are required for a more accurate balanced model. The findings are illustrated using several numerical examples.

11:22AM H19.00005 Performance and robustness comparison between reduced-plant-based and directly-reduced optimal control[1]

, KEVIN CHEN, CLARENCE ROWLEY, Princeton University — A common objective in flow control is the construction of a low-dimensional controller from a high-dimensional plant modeling the fluid dynamics. Traditionally, the controller is constructed from a reduced-order model of the original plant. Recent advances, however, have allowed the construction of the H_2 optimal controller directly from the full-dimensional plant; this controller could then be reduced to a lower dimension. This study explores these two methods of arriving at a low-dimensional optimal controller, using the Ginzburg-Landau equation as a testbed. The controller's performance is evaluated by the H_2 norm of the transfer function from state disturbances and sensor noise to costs on the state and input magnitude. The robustness is measured primarily by the coprime stability margin, for which analytic bounds are available via the ν-gap metric. The directly-reduced controller generally achieves greater performance and robustness than the reduced-plant-based controller, but takes longer to compute for large-dimensional plants. A decrease in performance and robustness is attributed to faster growth rates of an instability, greater non-normality or time lag, and lower model reduction order.

[1]This work was supported by the DOD NDESG.

11:35AM H19.00006 ABSTRACT WITHDRAWN —

11:48AM H19.00007 Comparison of two different approaches for the control of convectively unstable flows

, FABIEN JUILLET, PETER SCHMID, Ecole Polytechnique, BEVERLEY MCKEON, CalTech, PATRICK HUERRE, Ecole Polytechnique — The probably most widely used control strategy in the literature is based on the Linear Quadratic Gaussian (LQG) framework. However, this approach seems to be difficult to apply to some fluid systems. In particular, due to their high sensitivity to external noise, amplifier flows are hard to control and the classical LQG compensator may be unable to describe the noise with sufficient accuracy. Another strategy aims at directly measuring these noise sources through a sensor called "spy." The LQG and the spy approaches will be presented and compared using the Ginzburg-Landau equation as a model. It will be shown that the use of a spy is particularly relevant for convectively unstable systems. In addition, the ability of Subspace Identification Methods to provide satisfactory models is demonstrated. Finally, the findings from the Ginzburg-Landau investigation are generalized and applied to a more realistic system, namely a backward-facing step at $Re = 350$. Support from Ecole Polytechnique and the Partner University Fund (PUF) is gratefully acknowledged.

12:01PM H19.00008 Application of system-identification by ARMarkov and sensitivity analysis to noise-amplifier models

, NICOLAS DOVETTA, PETER SCHMID, Ecole Polytechnique, DENIS SIPP, ONERA-DAFE, BEVERLEY MCKEON, CalTech — Separated flow often exhibit amplification of external noise sources via an interaction with shear layer instabilities. In order to manipulate this amplification process we consider a data-based control design strategy. The first step is to build a state-space representation of the input-output transfer function. An auto-regressive representation is used that explicitly includes Markov parameters (ARMarkov). This is then coupled with the eigensystem realization algorithm (ERA) which yields a reduced-order state-space representation of the problem. In real experiments the data is contaminated by measurement noise or by non-linearities which are not accounted for by the present approach. In order to enforce robustness of the identification-realization procedure a sensitivity analysis of the algorithm is performed. These sensitivities provide quantitative criteria to find the most robust way of identifying the system using the ARMarkov/ERA algorithm. The system-identification and sensitivity framework will be demonstrated on the Ginzburg-Landau equation. Support from the Partner University Fund (PUF) is gratefully acknowledged.

12:14PM H19.00009 Estimation of the Proximity and Density Field of a Moving Gaseous Source Using a Sensing Aerial Vehicle[1] , NIKOLAOS GATSONIS, JEFF COURT, MICHAEL DEMETRIOU, WPI — This work considers the estimation of the density field arising from an unknown moving gaseous source in the troposphere using measurements obtained with a sensor onboard an aerial vehicle. The gas dispersion process is modeled by the advection-diffusion partial differential equation in 3d with spatially and time varying ambient mean velocity and eddy diffusivities. Our approach strongly couples the adaptive, multigrid numerical solution of the advection-diffusion PDE with the estimate of the process state (spatio-temporal density) as provided by the maxima of concentration localization estimation scheme. The guidance of the sensing aerial vehicle is dictated by the performance of the estimation scheme. Computational results demonstrate the effectiveness of the approach in estimating the density of the plume and the proximity of the gaseous source under realistic atmospheric conditions.

[1]This work was sponsored by the Air Force Office of Scientific Research, USAF, Dynamics and Controls Program, under grant/contract number FA9550-09-1-469

12:27PM H19.00010 Numerical Modeling and Simulations on Electo-Active Polymer Flow Control , ANDREW WEDDLE, MICHAEL AMITAY, LUCY ZHANG, Rensselaer Polytechnic Institute — The primary focus of this study is to identify the effects of vibrating Electro-Active Polymer (EAP) flow control on the flow field, specifically within the boundary layer. The EAPs represent a light-weight and adaptable flow control solution for micro-air vehicles (MAV). In this study, the interaction of the flow field over a flat plate and NACA 0009 airfoil are modeled at a Reynolds number of 20,000 using an Arbitrary Lagrangian Eulerian finite element formulation. In the simulations, the EAP vibration is prescribed based on the measurements from the experiments. The results show the EAPs do alter the boundary layer flow field and the size of the separation bubble. Three different diameter EAPs are examined on the flat plate model: 6mm, 9mm, and 12mm. Each is evaluated at different vibrational frequencies and maximum amplitudes. The performance of the EAPs on the NACA 0009 model are also evaluated while the airfoil is experiencing a pitching motion and gusts. Both instantaneous and time averaged flow fields are analyzed. The results from the numerical simulations are compared to baseline CFD simulations and wind tunnel results.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H20 Interfacial/Thin Film Instability IV 323

10:30AM H20.00001 Surfactant spreading on a thin liquid film: testing models with experiments[1] , MICHAEL SHEARER, NC State University, ELLEN PETERSON, Carnegie-Mellon University, ANTOINE LE BOUIL, University of Rennes, KAREN DANIELS, NC State University — We report on recent experiments with NBD-PC lipid spreading on a thin layer of glycerin, in which small concentrations of molecules are visualized using fluorescence. The varying profile of the film surface is simultaneously captured using a laser line. In lubrication models, insoluble surfactant is passively transported by the free surface. This assumption is plausible in the regime of small surfactant concentrations, for which the spreading molecules form a monolayer, but is questionable for larger quantities. We record the time-dependent distribution of surfactant for various initial concentrations down to a monolayer, making direct comparisons between the experimental results and predictions from simulations of the model system of partial differential equations.

[1]Support from NSF-FRG grant DMS-0968258.

10:43AM H20.00002 The size and shape of gas-focused viscous micro-jets , C. FERRERA, University of Extremadura, A.M. GANAN-CALVO, University of Seville, J.M. MONTANERO, E.J. VEGA, University of Extremadura, M.A. HERRADA, University of Seville — The size and shape of gas-focused viscous micro-jets are analyzed theoretically and experimentally. These micro-jets are shaped by the action of a co-flowing gas stream due to both the pressure drop in the axial direction occurring in front of the discharge orifice, and the tangential viscous stress caused by the difference between the velocities of the gas and jet behind the orifice. The slender approximation is used to describing the shape of the tapering meniscus and the emitted liquid ligament. Assuming that the driving force takes a uniform value over the entire liquid domain, a universal (self-similar) solution of the momentum equation can be obtained. Experiments were conducted to assess the validity of that solution for a wide range of liquid viscosities. A remarkable collapse into a single curve is obtained for of all jet diameters measured beyond the orifice. This result shows that the driving force mentioned above attains a rather homogeneous value at the region where the micro-jet develops. The universal solution also provides satisfactory results in front of the orifice for sufficiently slender liquid meniscus, provided that the ratio capillary-to-orifice distance to orifice diameter takes sufficiently small values. The approach used in this work can also be applied to study other microjet generation means (co-flowing, electrospray, electrospinning. . .).

10:56AM H20.00003 Air entrainment by viscous contact lines , TAK SHING CHAN, University of Twente, ANTONIN MARCHAND, BRUNO ANDREOTTI, PMMH, ESPCI, Univ. Paris-Diderot, JACCO H. SNOEIJER, University of Twente — We study the entrainment of air by moving contact lines, by plunging a solid plate into a reservoir of silicone oil. Above a critical plunging velocity we observe the formation of an air film, typically of 10 microns thickness, which subsequently destabilizes to form bubbles. The breakup of the air film occurs through the nucleation of "rewetting" holes, growing regions of liquid reestablishing contact with the surface – the inverse process of the classical dewetting holes. Typical speeds for air entrainment turn out to be orders of magnitude larger than the speed for liquid film deposition. By varying the viscosity of the silicone oil, it is found that this critical speed depends on properties of both the liquid and the air. We propose an approximate hydrodynamic model that accounts for this two-phase nature of air entrainment.

11:09AM H20.00004 Jetting-to-bubbling transition in planar coflowing air-water sheets[1] , R. BOLAÑOS-JIMÉNEZ, Universidad de Jaen, Spain, A. SEVILLA, Universidad Carlos III de Madrid, Spain, C. GUTIÉRREZ-MONTES, E. SANMIGUEL-ROJAS, C. MARTÍNEZ-BAZÁN, Universidad de Jaen, Spain — We study the transition between the two regimes experimentally observed when a plane air sheet surrounded by a coflowing water sheet discharges into stagnant air: a bubbling regime, which leads to the periodic break-up of the air sheet, and a jetting regime, where both sheets evolve slowly downstream without breaking. A jetting-to-bubbling transition curve has been experimentally obtained for a fixed liquid-to-gas thickness ratio, $h = h^*_{w,0}/h^*_{a,0} \simeq 5.27$, as a function on the values of two control parameters, namely, the Weber number, $We = \rho_w \, u^{*2}_{w,0} \, h^*_{a,0}/\sigma$, and the velocity ratio, $\Lambda = u^*_{w,0}/\bar{u}^*_{a,0}$, where $u^*_{w,0}$ and $\bar{u}^*_{a,0}$ are the water velocity and the mean air velocity at the exit slit, respectively, and $h^*_{a,0}$ and $h^*_{w,0}$ are the half-thicknesses of the air and water sheets at the exit. A linear spatiotemporal stability analysis, contemplating the downstream evolution of the sheets, shows good agreement with the experiments if a sufficiently long region of absolute instability is postulated.

[1]Supported by the Spanish MICINN, Junta de Andalucía and EU Funds under projects DPI2008-06624-C03-02 and P07-TEP02693.

11:22AM H20.00005 Nonlinear instabilities and Koopman Modes in axisymmetric multiphase shear flows , DANIEL DUKE, JULIO SORIA, DAMON HONNERY, Monash University, LABORATORY FOR TURBULENCE RESEARCH IN AEROSPACE & COMBUSTION TEAM[1] — The shear-driven atomisation of multiphase flows is a process about which relatively little is understood, as it is driven by complex, detailed and short-lived transient instabilites. Novel high-resolution surface velocimetry of axisymmetric annular sheets reveals new detail which has not been previously observed through spatially and temporally resolved velocity profiles. We hypothesise that three simultaneous and separate physical sources of instability may explain the complexity of this flow. A novel application of the Hilbert Transform permits the isolation of these distinct instabilities. The first two are the more well-known nozzle boundary layer and free shear layer driven instabilities. The third is due to nonlinear sheet distortion leading to rupture and may be directly driving the primary atomisation process. This instability has not been previously observed in isolation and is inherently non-linear and non-sinusoidal which has made its eduction challenging. A novel application of Koopman modes through use of the Dynamic Mode Decomposition suggests that traditional theoretical approaches to understanding these flows may need to be re-examined.

[1]Dept. Mechanical & Aerospace Engineering, Monash University

11:35AM H20.00006 Stability of flow focusing: The minimum attainable flow rate , J.M. MONTANERO, N. REBOLLO, A. ACERO, C. FERRERA, University of Extremadura, M.A. HERRADA, A.M. GANAN-CALVO, University of Seville — We analyze both theoretically and experimentally the stability of the steady jetting regime reached when liquid jets are focused by coaxial gas streams. In the low-viscosity case, viscous dissipation in the feeding capillary and liquid meniscus seem to be the origin of the instability. For high-viscosity liquids, the breakdown of the jetting regime takes place when the pressure drop cannot overcome the resistance force offered by surface tension. The characteristic flow rates for which the tapering menisci become unstable do not depend on the pressure drop applied to the system to produce the micro-jet. They increase (decrease) with viscosity for very low (high) viscosity liquids. Experiments confirmed the validity of the above conclusions. For each applied pressure drop, there is a minimum liquid flow rate below which the liquid meniscus drips. The minimum flow rates become practically independent of the applied pressure drop for sufficiently large values of this quantity. There exists an optimum value of the capillary-to-orifice distance for which the minimum flow rate attains a limiting value, which constitutes the lowest flow rate attainable with a given configuration in the steady jetting regime. A two-dimensional stability map with a high degree of validity is plotted on the plane defined by the Reynolds and capillary numbers based on the limiting flow rate.

11:48AM H20.00007 Ring-waves dominate mass transport in air-driven core-annular flows , JEFFREY OLANDER, ROBERTO CAMASSA, GREG FOREST, UNC, LONG LEE, University of Wyoming, JOHN MELLNIK, H. REED OGROSKY, UNC — To better understand the movement of mucus through the trachea that arises as a result of air flow, we perform a series of experiments to emulate mucus movement by an air-driven vertical flow of high- viscosity silicone oil through a thin glass tube. When a constant flux of air is delivered through the bottom of the tube, instabilities arise, generating upward moving waves at the oil/air interface. We present experiments and theory that demonstrate that these waves are actually vortex ring waves of annular fluid rolling up the tube between the liquid substrate layer and gas core. A long-wave asymptotic model is developed that exhibits this mass- transport wave behavior. The region of parameter space where such mass transport waves exist is found. In addition, wavelength, wave speed, and mean thickness of the oil lining the tube are found as a function of air speed. A comparison is made between the model and the experiments. These results give insight into the clearing of mucus in the trachea by air flows.

12:01PM H20.00008 Surface tension gradient-induced gel fracture[1] , CONSTANTINE SPANDAGOS, PAUL LUCKHAM, OMAR MATAR, Imperial College London — The spreading of surfactant-laden droplets on the surface of gels can be accompanied by the fracture of the gel surface and the subsequent formation of crack-like patterns. This intriguing phenomenon has been observed on the surface of different types of gels, and for the spreading of both convectional and superspreading surfactants. Understanding the mechanism behind this crack formation is believed to contribute significantly to the control and improvement of a large number of processes that involve systems of spreading liquids and gel-like materials. It is suggested that Marangoni stresses generated by surface tension gradients between the surfactant and the gel are responsible for the gel surface fracture. The gel seems to "unzip" in a direction perpendicular to that of the crack propagation. Furthermore, our systematic study reveals that a crack can be formed only within the experimental conditions that allow $S/\Delta w$ to be greater than G', where S is the surface tension difference between the surfactant and the underlying gel, Δw is the change of the width of a crack and G' is the storage modulus of the substrate.

[1]EPSRC Grant number EP/E046029/1

12:14PM H20.00009 Investigation of partially dewetting flow at Re$\sim$O(100) and Ca$\sim$O(0.01) , HYOUNGSOO KIM, CHRISTIAN POELMA, JERRY WESTERWEEL, Delft University of Technology — We study a free surface flow problem with moving contact lines at a Reynolds number Re$\sim$O(100) and a capillary number Ca$\sim$O(0.01). This problem corresponds to the flow in an immersion drop applied in a liquid-immersion lithography machine; so-called a shear-driven confined liquid layer. At a certain critical condition, droplet break-up occurs. We quantitatively investigate this free surface flow problem by means of shadowgraphy measurement and tomographic PIV. For increasing shear rates, we observe four successive regimes at the rear end of the liquid layer. These regimes are identified based on the shape of the elongated tail and denoted as oval, corner, cusp, and pearling. Interestingly, the same qualitative behavior is also observed in a different geometry in the Stokes flow regime (Podgorski et al. 2001). Moreover, additional quantitative similarities are found by evaluating analytical results for sliding drops, based on Stokes flow: the Cox-Voinov model, the three-dimensional lubrication model and a self-similar flow pattern. Surprisingly enough, these results match with the observations even for the present intermediate Reynolds number free surface flow. To understand the underlying mechanism behind this agreement, we have performed a scale analysis based on the momentum equations. Finally, we present the flow structure in this problem is expressed in terms of the results of the lubrication equation.

12:27PM H20.00010 Oscillatory bubbles induced by geometrical constraint[1] , ANNE JUEL, MICKAEL PAILHA, ANDREW HAZEL, University of Manchester — Microscale process engineering requires precise control of bubbles and droplets. We investigate geometry-induced control and find that a centered constriction in the cross section of rectangular tubes can lead to new families of steadily propagating bubbles, which localize in the least-constricted regions of the cross section. Tuning the constriction geometry can cause a switchlike transition from centered to localized bubbles at a critical value of the flow rate: a mechanism for flow-rate-driven bubble control [1]. Striking and robust periodic oscillations develop on the advancing air-fluid interface that can dramatically reduce the volume of fluid extracted. The dynamics of the oscillations are consistent with their arising from a global homoclinic connection between the stable and unstable manifolds of a steady, symmetry-broken solution.

[1] A. de Lózar, A. Heap, F. Box, A.L. Hazel & A. Juel *Partially-occluded tubes can force switch-like transitions in the behavior of propagating bubbles*, Phys. Fluids. **21**, 101702; doi: 10.1063/1.3247879, 2009.

[1]Funded by EPSRC.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H21 Vortex Dynamics IV 324-325

10:30AM H21.00001 Vorticity Dynamics in Actuated Transverse Jets , FABRICE SCHLEGEL, AHMED F. GHONIEM, MIT — Transverse jets are important to various industrial applications (film cooling, primary or dilution jets in gas turbines). Our previous results on the impact of the boundary layer detachment on the jet evolution show the sensibility of the overall jet dynamics to the near-nozzle conditions. Small perturbations at the nozzle exit are thus expected to act as powerful tools for control of the jet trajectory, its spanwise spreading, and its mixing properties. In this study, we demonstrate actuation strategies that manipulate the jet via nozzle edge perturbations, helical perturbations and the addition of delta-tabs at the nozzle exit.

10:43AM H21.00002 Close relative equilibria of identical point vortices[1] , TOBIAS DIRKSEN, Technical University of Denmark, HASSAN AREF, Virginia Tech — Via numerical solution of the classical problem of relative equilibria for identical point vortices on the unbounded plane we have found configurations that are very close to the analytically known, centered, symmetrically arranged, nested equilateral triangles. Numerical solutions of this kind were found for $3n+1$ vortices, where $n = 2, 3, \ldots, 30$. A sufficient, although apparently not necessary, condition for this phenomenon of close solutions is that the "core" of the configuration is marginally stable, as occurs for a central vortex surrounded by an equilateral triangle. The open, regular heptagon also has this property, and new relative equilibria close to the nested, symmetrically arranged, regular heptagons have been found. The centered regular nonagon is also marginally stable. Again, a new family of close relative equilibria has been found. The closest relative equilibrium pairs occur, however, for symmetrically nested equilateral triangles. The numerical evidence is surveyed and related recent work mentioned. A Letter in *Physics of Fluids* **23** (2011) 051706 is available.

[1]Supported in part by the Danish National Research Foundation through a Niels Bohr visiting professorship.

10:56AM H21.00003 Stability of trailing vortices with radial stratification , JEROME FONTANE, LAURENT JOLY, AURIANE AUDOUIN, Université de Toulouse, ISAE, DAEP — We look at the effects of the radial density stratification on the stability of the q-vortex, a commonly accepted model for aircraft trailing vortices. It has been demonstrated that the 2D Lamb–Oseen vortex develops a Rayleigh–Taylor instability when its core is heavier than the surrounding fluid (Joly, Fontane & Chassaing 2005, Sipp *et al* 2005). The underlying mechanism relies on baroclinic vorticity generation due to any misalignment between the density gradient and the centripetal acceleration field. The instability is triggered provided that the density decreases radially somewhere in the vortex core. This mechanism is also active in the 3D trailing vortex and affects its stability characteristics due to the addition of an axial component in the acceleration field. We show that the unstable center modes of the homogeneous case (Fabre & Jacquin 2004) are promoted in a q-vortex with a heavy core. Their growth rate increases while their m-spiral structure is preserved. For an Atwood number $At = 0.5$, their predicted growth rate can be ten times the ones found in the homogeneous case. Furthermore, the unstable domain is extended far beyond the neutral curve in the homogeneous case, with unstable modes observed for Swirl numbers up to $q = 5$. It is argued here that corresponding density perturbations could eventually lead to the development of new and original strategies to decrease the lifespan of aircraft trailing vortices and greatly reduce their unwanted side-effects on contrails persistence and air traffic regulations.

11:09AM H21.00004 Linear response of a vortex column - singular eigenfunctions and growth mechanisms , ANUBHAB ROY, Jawaharlal Nehru Centre for Advanced Scientific Research, Bangalore, ABHISHEK S, Indian Institute of Technology, Madras, HARISH N. DIXIT, University of British Columbia, Vancouver, GANESH SUBRAMANIAN, Jawaharlal Nehru Centre for Advanced Scientific Research, Bangalore — A vortex column supports oscillations known as Kelvin modes, eigenmodes that are irrotational outside the vortex core. In order to understand the interaction of a vortex with an external vortical disturbance field, we use an extended modal description that incorporates a singular continuous spectrum with eigenmodes that are vortical outside the core. The continuous spectrum eigenfunctions are explicitly evaluated for a Rankine vortex; the description is extended to smooth vorticity profiles based on an analogy with stratified shear flows. Next, in the framework of an initial value problem, we analyze the inviscid resonant interaction between a vortex column and suitably localized initial conditions. It is shown that while a Rankine vortex allows for an unbounded secular growth in response an infinitely localized initial condition, smooth vorticity profiles, with a non-zero critical layer vorticity gradient, exhibit a saturation resulting from a perturbation-vorticity-induced screening mechanism. The effects of an upstream tilt on this novel saturation response are investigated.

11:22AM H21.00005 Inviscid linear stability of a Vatistas-trailing line vortex , HOSSEIN ABDOLLAHI, Concordia University, HAMID AIT ABDERRAHMANE, GEORGIOS H. VATISTAS, Concordia University — The paper examines the inviscid linear stability of a Vatistas-trailing line vortex using normal mode analysis. The employed numerical method is based on the Riccatti transformation. The stability of Vatistas-trailing line vortex is discussed with respect to the well-known Batchelor's trailing line vortex. The results show that Vatistas' vortex is more stable.

11:35AM H21.00006 On Existence of Relative Equilibria of Two–Dimensional Vortex Sheets[1] , BARTOSZ PROTAS, McMaster University, MARCEL RODNEY, McGill University — In this study we consider the existence of relative equilibria of two–dimensional vortex sheets. We focus on open sheets and derive conditions which must be satisfied by equilibrium configurations of such sheets. It is shown that, in contrast to the time–dependent case, such sheets must be everywhere orthogonal to the velocity field of the coordinate system in which they are stationary. Finally, we provide a rigorous demonstration that for vortex sheets arising from desingularization of translating (counter–rotating) and corotating pairs of point vortices such equilibrium configuration do not in fact exist. The argument is based on classical results concerning existence of solutions of singular integral equations.

[1]Supported by NSERC (Canada) Discovery and USRA Programs.

11:48AM H21.00007 Stability of a helical vortex tube with axial flow , YUJI HATTORI, IFS, Tohoku University, YASUHIDE FUKUMOTO, IMI, Kyushu University — The stability of a helical vortex tube with axial flow is studied analytically. The base flow is obtained by solving the Euler equation perturbatively assuming small ratio of core to curvature radius, which is denoted by ε, and Rankine vortex with uniform axial flow at the leading order. We apply both local and modal stability analysis. By local stability analysis we show that the flow is subject to not only curvature instability but also Coriolis instability, both having the same resonance condition. The unstable growth rate is $O(\varepsilon)$ and given by the magnitude of a sum of the complex numbers corresponding to the two instabilities. Combined effects of the axial flow and the torsion of the helical vortex tube appear as $O\left(\varepsilon^2\right)$ modification. These results are confirmed by the modal stability analysis.

12:01PM H21.00008 Inertial dynamics of chains: slack, stress, and convective instabilities , JAMES HANNA, CHRISTIAN SANTANGELO, UMass-Amherst — Inertial chains may be thought of as one-dimensional incompressible/inextensible fluids or solids moving in three dimensions. Incompressibility is enforced by a stress screened by the chain's curvature (slack). The nature of the stress— tensile or compressive, uniform or spatially varying— governs the stability of the motion. The most stable motions, characterized by a uniform tensile stress, belong to a wide class that includes travelling waves of curvature and torsion. Convective instabilities exist in the presence of stress gradients; we present a striking example from a tabletop experiment involving a growing arch in a straightening chain. This work adds to a large body of literature on locally arc length preserving dynamics of curves arising in the study of thin objects such as elastic rods, vortex filaments, and oceanic jets.

12:14PM H21.00009 The bifurcation and stability of vortex flow states at near-crtical swirl ratios , LEI XU, ZVI RUSAK, RPI, SHIXIAO WANG, Auckland University — A theoretical and computational study of the bifurcation and stability of vortex flow states at near-critical swirl levels is presented. The theoretical analysis is based on a nonlinear model problem of the dynamics of small perturbations on a columnar flow. The model is first used to study the bifurcation of additional branches of equilibrium states and the stability of these states. The response of the columnar flow to various initial conditions is also studied. A frequency response analysis to inlet perturbations at various amplitudes and initial conditions is conducted and demonstrates the ability to control the growth of the perturbations when certain oscillations are imposed. However, this approach is limited to relatively small-amplitude initial perturbations. A bang-bang control approach where swirl is changed as a function of outlet centerline axial speed shows the ability to induce limit-cycle oscillations on the flow and control the growth of perturbations at a wider range of swirl levels and initial conditions.

12:27PM H21.00010 The global nonlinear stability of a swirling flow in a pipe , ZVI RUSAK, RPI, SHIXIAO WANG, Auckland University, LEI XU, RPI, STEVE TAYLOR, Auckland University — The dynamics of a perturbed axisymmetric, near-critical swirling flow in a long, finite-length, straight, circular pipe is studied through a weakly nonlinear analysis. The flow is subjected to non-periodic inlet and outlet conditions. Examples of flow dynamics at various near-critical swirl levels in response to different initial perturbations demonstrate the important role of the nonlinear steepening terms in perturbations' dynamics. Results reveal the evolution of explosive, faster-than-exponential, shape-changing modes as perturbations grow into a vortex breakdown process. Further analysis of the model problem shows the important role of the nonlinear evolution in (i) the transfer of perturbation's kinetic energy between the boundaries and flow bulk, (ii) the evolution of perturbations in practical concentrated vortices, and (iii) the design of control methods to stabilize vortex flows. A robust feedback control method to stabilize a solid-body rotation flow in a pipe at a wide range of swirl levels above critical is developed. Applicability of this method to stabilize medium and small core-size vortices is also discussed.

Monday, November 21, 2011 10:30AM - 12:14PM –
Session H23 Non-Newtonian Flows I 326

10:30AM H23.00001 Dynamics of associative polymer solutions: Capillary break-up, jetting and rheology , VIVEK SHARMA, Hatsopoulos Microfluids Laboratory, Dept of Mechanical Eng., MIT, Cambridge, MA 02139, JAMES G. SERDY, Laboratory of Manufacturing and Productivity, MIT, PHIL THREFALL-HOLMES, AkzoNobel, UK, GARETH H. MCKINLEY, Hatsopoulos Microfluids Laboratory, Mechanical Eng., MIT, Cambridge, MA 02139 — Associative polymer solutions are used in extensively in the formulations for water-borne paints, food, inks, cosmetics, etc to control the rheology and processing behavior of multi-component dispersions. These complex dispersions are processed and used over a broad range of shear and extensional rates. Furthermore, the commercially relevant formulations use dilute solutions of associative polymers, which have low viscosity and short relaxation times, and hence their non-Newtonian response is not apparent in a conventional rheometer. In this talk, we explore several methods for systematically exploring the linear and nonlinear solution rheology of associative polymer dispersions, including: fractional model description of physical gelation, high frequency oscillatory tests at frequencies up to 10 kHz, microfluidic shear rheometry at deformation rates up to 1000000 /s and the influence of transient extensional rheology in the jet breakup. We show that high deformation rates can be obtained in jetting flows, and the growth and evolution of instability during jetting and break-up of these viscoelastic fluids shows the influence of both elasticity and extensibility.

10:43AM H23.00002 Designing a simple rheometer: Unsteady dam-break flows of power-law fluids , LAURA CHILDS, ANDREW J. HOGG, University of Bristol — Many geophysical flows, such as mud slides, debris flows and avalanches, are non-Newtonian, and consequently exhibit complex flow behaviour. This is often due to an underlying microstructure within the flow – for example, a suspension of particles within an interstitial fluid. We develop a method for determining the rheological parameters of a power-law fluid through the utilization of both numerical results and data from simple laboratory experiments. The model employed describes an unsteady dam-break flow of a viscous material within a rectangular channel, capturing the shape of the free surface and the streamwise velocity; a novel feature of this work is that it accounts for the effect of the containing side walls of a channel of any specified dimensions. Much of the previous work in this area has neglected this aspect of the physics, but here it can be shown that the side walls have an appreciable influence on the flow. The method relies on the numerical calculation of the flux through a cross-section of the channel, which has been done using both a finite element solver and a pseudo-spectral method, coupled to an evolution equation along the axis of the channel. Results from laboratory experiments will be shown in order to demonstrate the application, and effectiveness, of the method.

10:56AM H23.00003 Rheology of Concentrated Colloidal Suspensions in the Approach of the Glass Transition , HORST WINTER, NSF, MIRIAM SIEBENBUERGER, MATTHIAS BALLAUFF, Helmholtz-Zentrum Berlin, Germany — Concentrated, non-crystallizing colloidal suspensions in their approach of the glass state exhibit distinct dynamics patterns. This is demonstrated with a model suspension at increased volume fraction of solid. The glass is defined by arrested motion of the spherical suspension particles. Dynamic mechanical experiments suggest a powerlaw rheological constitutive model for near-glass viscoelasticity as presented here. The rheological parameters used for this model originate in the Mode-Coupling Theory. The proposed constitutive model provides explicit expressions for the steady shear viscosity, the steady normal stress coefficient, the modulus-compliance relation, and the α-peak of G". The relaxation pattern distinctly differs from gelation.

11:09AM H23.00004 Selective withdrawal as a method to measure extensional viscosity , E. TREJO, R. ZENIT, Universidad Nacional Autonoma de Mexico, J.J. FENG, University of British Columbia — The extensional viscosity of a liquid is a property that has shown to be very difficult to measure experimentally. We propose the use of the so-called selective withdrawal device to measure it. A viscous fluid layer is withdrawn from below with a tube. The suction deforms the free surface and generates an extensional ow at the entrance of the pipe. We conducted measurements of the extensional rate using PIV and related its value with the deformation of the free surface to determine the extensional viscosity. Some preliminary results will be presented and discussed for both Newtonian and viscoelastic liquids.

11:22AM H23.00005 Finite and Infinite Width Stokes Layers in a Power-Law Fluid , STEPHEN WILSON, DAVID PRITCHARD, CATRIONA MCARDLE, University of Strathclyde — Self-similar solutions for the oscillatory boundary layer (the "Stokes layer") in a semi-infinite power-law fluid bounded by an oscillating wall (the so-called Stokes problem) are obtained and analysed. These semi-analytical solutions differ qualitatively from the classical solution for a Newtonian fluid, both in the non-sinusoidal form of the velocity oscillations and in the manner at which their amplitude decays with distance from the wall. In particular, for shear-thickening fluids the velocity reaches zero at a finite distance from the wall, and for shear-thinning fluids it decays algebraically with distance, in contrast to the exponential decay for a Newtonian fluid. We demonstrate numerically that these self-similar solutions provide a good approximation to the flow driven by a sinusoidally oscillating wall. Further details can be found in the recent paper by D. Pritchard, C. R. McArdle and S. K. Wilson entitled "The Stokes boundary layer for a power-law fluid," in *Journal of Non-Newtonian Fluid Mechanics* **166**, 745–753 (2011).

11:35AM H23.00006 Britle failure of non-Newtonian, floating, extensional flows[1] , ROIY SAYAG, MICHAEL WORSTER, DAMTP, University of Cambridge — Glacier ice is driven by gravity to flow from the land, where it is under shear, into the ocean, where it floats and extends. Owing to its non-Newtonian rheology, the ice can flow axisymmetrically over the bed but undergo brittle failure once it is floating on the ocean, as observed for example in crevassing of ice shelves. We model this coupled flow as an intrusion of a viscous gravity current into a denser ocean and study it both theoretically and experimentally. We have conducted laboratory experiments using a shear-thinning suspension that represents ice, and a denser inviscid fluid that represents an ocean. The non-Newtonian fluid was released at a constant flux through a cylindrical nozzle over a horizontal plane. The grounded, shear-dominated region of the flow was axisymmetric throughout the experiment, while past the transition line axisymmetry broke down into a seemingly ordered set of finger-like extensions (floating shelves) that demonstrated brittle behaviour. We have found that the width of the fingers as well as their radial extent increase with the flux. We attempt to explain these observations through a fingering instability that is driven by the dynamical differences between the two flow domains and by the material rheology, and we project that analysis to formulate a linkage between the material properties of ice and an upper bound on the width of ice shelves.

[1]NERC

11:48AM H23.00007 Numerical Analysis of Gravity-driven Spreading of Viscoelastic Fluids: Investigation of the Effect of Shear-thinning and Elastic Behavior , BIN HU, SARAH KIEWEG, University of Kansas — Many complex fluids of interest exhibit viscoelastic hehavior. Polymeric drug delivery vehicles, such as anti-HIV topical microbicides, are among these fluids. For the optimal design of microbicides, the combined effect of shear-thinning and elastic behavior on the gravity-driven spreading of viscoelastic fluids is studied. We develop a 2D model to simulate the fluids spreading down an incline using ANSYS POLYFLOW software package. Arbitrary Lagrangian-Eulerian (ALE) method combined with Lagrangian remeshing is applied to track the moving free surface of fluids during spreading. Adaptive meshing method is used to generate high quality mesh for the remeshing process. Based on an elastic viscous split stress (EVSS) approach, several differential viscoelastic constitutive models are studied to investigate the combined effect of shear-thinning and elastic behavior. Mesh convergence test and constant volume check are studied to verify the new model. Moreover, the new model with zero elasticity is compared with previous studies of Newtonian and power-law fluids.

12:01PM H23.00008 Heat Transfer and Couette Flow of a Chemically-Reacting Non-Linear Fluid[1] , KEREM UGUZ, Bogazici University, Department of Chemical Engineering, Istanbul, Turkey, MEHRDAD MASSOUDI, U.S. Department of Energy, National Energy Technology Laboratory (NETL), Pittsburgh, PA, 15236, USA — The velocity, temperature, and concentration fields of a reacting fluid flowing in a channel are obtained. The fluid is assumed to flow between two parallel plates where one of the plates is sheared at constant speed whereas the other one is stationary. The plates are kept at constant hot and cold temperatures. Chemically reacting fluids are used in many industries and technologies such as combustion, catalysis, and biological systems and they usually show non-Newtonian behavior. The fluid is assumed to obey generalized power-law constitutive equation and its physical properties, i.e., viscosity, thermal conductivity and the diffusion coefficients are assumed to be a function of the concentration. The system is studied numerically for various parameters.

[1]Kerem Uguz acknowledges support from Bogazici University Research Fund Project 6007.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H24 Convection and Buoyancy Driven Flows I 327

10:30AM H24.00001 Buoyancy driven acceleration in a hospital operating room indoor environment[1] , JAMES MCNEILL, JEAN HERTZBERG, JOHN ZHAI, University of Colorado, Boulder — In hospital operating rooms, centrally located non-isothermal ceiling jets provide sterile air for protecting the surgical site from infectious particles in the room air as well as room cooling. Modern operating rooms are requiring larger temperature differences to accommodate increasing cooling loads for heat gains from medical equipment. This trend may lead to significant changes in the room air distribution patterns that may sacrifice the sterile air field across the surgical table. Quantitative flow visualization experiments using laser sheet illumination and RANS modeling of the indoor environment were conducted to demonstrate the impact of the indoor environment thermal conditions on the room air distribution. The angle of the jet shear layer was studied as function of the area of the vena contracta of the jet, which is in turn dependent upon the Archimedes number of the jet. Increases in the buoyancy forces cause greater air velocities in the vicinity of the surgical site increasing the likelihood of deposition of contaminants in the flow field. The outcome of this study shows the Archimedes number should be used as the design parameter for hospital operating room air distribution in order to maintain a proper supply air jet for covering the sterile region.

[1]This work is supported by ASHRAE

10:43AM H24.00002 Particle-Laden Viscous Gravity Currents , SANDEEP SAHA, LAURENT TALON, Lab. FAST, CNRS, DOMINIQUE SALIN, UPMC 6, Lab. FAST CNRS, POROUS MEDIA TEAM — The extension of a gravity current in lock-exchange problems, proceeds as square root of time in the viscous regime. In the presence of particles, however, this scenario is drastically altered due to sedimentation in a manner similar to the well- known Boycott effect. The spreading of particle-laden gravity currents is investigated with numerical simulations based on a Lattice-Boltzmann method. The settling of particles is modelled using a flux function for capturing sudden discontinuities in particle concentration travelling as kinematic shock waves. Contrary to conventional gravity currents, sedimentation leads to the formation of two additional fronts: a horizontal front descending vertically and a sediment layer that ascends as the particles accumulate. Two regimes emerge in the spreading process: the latter corresponding to the lateral advance of the sediment deposit and the former characterised by the vertical motion of the two fronts. An increase in the initial concentration hastens the time at which the regime change occurs and impedes the overall spreading process. The sedimentation velocity of the particles either slows down or speeds up the edges of the gravity current. A model based on lubrication theory is derived to explain the results and identify scaling laws.

10:56AM H24.00003 Self-consistent definition for the variable depth of unsteady, turbulent gravity currents , HAFIZ JUNAID ANJUM, JIM MCELWAINE, DAMTP, University of Cambridge, C.P. CAULFIELD, BPI and DAMTP, University of Cambridge — We used the data from two-dimensional direct numerical simulations of Boussinesq gravity currents to define a self-consistent depth, h, and reduced gravity, $g' = \frac{g}{\rho_0}(\rho - \rho_0)$, for the current in terms of moments of the current density field. We demonstrate that using these definitions to calculate the Froude number, $\mathrm{Fr}=u/\sqrt{g'h}$, gives a constant Froude number in constant-velocity and self-similar regime. At sufficiently high Reynolds number, our results are consistent with previous experimental and theoretical models (Shin *et al.* 2004, Nokes *et al.* 2008). We also develop a simple model to quantify the loss of mass from the gravity current head due to shear-induced vortices which propagate away from and behind the head.

Shin, J. O., Dalziel, S. B., Linden, P. F. (2004). Gravity currents produced by lock exchange. *J. Fluid Mech.* **521**,1-34.
Nokes, R.I., Davidson, M.J., Stepien, C.A., Veale, W. B., Oliver, R.L. (2008). The front condition for intrusive gravity currents. *J. Hydraul. Res.* **46** (6) 788-801.

11:09AM H24.00004 Laminar-turbulent cycles in strongly-confined lock-exchange flows , YUKIE TANINO, Imperial College London, FREDERIC MOISY, JEAN-PIERRE HULIN, FAST Laboratory, CNRS, Pierre & Marie Curie - Paris 6 and Paris-Sud 11 Universities — It is well established that certain shear flows display a transitional regime in which they alternate regularly between laminar and turbulent states. Here, we consider laminar- turbulent cycles in strongly-confined lock-exchange flows in an inclined tube. Simultaneous measurements of density and velocity fields using laser-induced fluorescence and particle image velocimetry demonstrate that the axial velocity evolves in a distinctive ramp-cliff pattern during each laminar-turbulent cycle. This pattern indicates that the flow accelerates as it relaminarizes, then decelerates rapidly with the breakdown of Kelvin-Helmholtz billows. In contrast to conventional interpretation, a single value of the Richardson number does not distinguish flows that subsequently exhibit turbulence from flows that do not. While the density contrast drives the shear flow macroscopically, local stratification does not directly control the onset of instability within experimental conditions. Instead, the measurements suggest that a necessary criterion for a ramp-cliff laminar-turbulent cycle in this flow configuration is for the local Reynolds number to exceed 2200-2300.

11:22AM H24.00005 Heat transport in a tilted channel[1] , XAVIER RIEDINGER, JEAN-CHRISTOPHE TISSERAND, FRANCESCA CHILLA, BERNARD CASTAING, ENS Lyon CNRS — We study heat convection in a channel between two chambers, the hot one being lower than the cold one. The channel is tilted with an angle varying up to 50 degrees and the power applied, to heat the hot chamber, varying up to 80 Watts. We perform both thermal and velocity measurements, through a PIV technique. We define a Nusselt number and a Rayleigh number according to the measured thermal gradient inside the channel. We show how increases the Nusselt number with the angle of the channel. Otherwise, the evolution of the thermal gradient shows four different regimes depending of the tilt angle and the power. When increasing power, the fluid flow changes from laminar to turbulent. For the turbulent case, we give a scaling law for the mean axial velocity. For intermediate parameters we differentiate another soft turbulent regime for which the velocity scaling law still apply but where the thermal gradient evolves much slowly with power. PIV measurements allow us to illustrate the transition from the laminar flow to the turbulent one and to distinguish a fourth regime charaterized by the destabilisation of laminar lateral jets.

[1]This work has been supported by the Agence Nationale pour la Recherche under the contract ANR-07-BLAN-0181.

11:35AM H24.00006 Effect of rate of change of external conditions on the transition from abnormal to normal steady-state convection , ALBERT SHARIFULIN, Perm State Technical University, ANATOLY POLUDNITSIN, Perm State University — Steady-state thermal convection regimes in a closed cavity undergo bifurcation on smooth variation of the external parameters. Bifurcations originating from changes in the Rayleigh number and angle of cavity inclination are investigated in the present work. We experimentally investigate the influence of the crossing rate through bifurcation curve on the transition process. The experiments were conducted with tilted cubic cavity filled with air, as described in [1], for fixed values of the Rayleigh number and different speeds of the slope of the cavity. It was found that at low velocities the slope of the rate of inclination does not affect the critical angle, beyond which is about the spontaneous transition from the anomalous convection mode to normal one. With no low-speed tilting the region of existence of the anomalous flow is sufficiently increased, and the transition process, as well as at low speeds, the slope is in rapid rotation around the vertical axis of the mass of air that fills the cavity. Possibility of applications to atmospheric behavior explanation and to Earth's mantle one is discussed.

[1] A.N. Sharifulin, A.N. Poludnitsin A.N., A.S. Kravchuk Laboratory Scale Simulation of Nonlocal Generation of a Tropical Cyclone. Journal of Experimental and Theoretical Physics, 2008, Vol.107, No.6, pp.1090-1093.

11:48AM H24.00007 Convection, thermal stratification and the energy penalty of localised heating with and without thermal mass , ANDREA KUESTERS, ANDREW W. WOODS, BP Institute, University of Cambridge — When a large enclosed space is heated from a point source, the space becomes vertically stratified in temperature. We develop a model for this stratification in the case of constant mixing ventilation, in which there is high level supply and extract of air, testing our model predictions with some new laboratory experiments. We demonstrate that with large ventilation flow or a small heat flux a strong two layer stratification develops in the space, whereas with low ventilation flux or large heat supply the stratification is weak. We explore how thermal mass affects this stratification and generalise the results to allow for a combination of a space with a distributed and a point source of heating. One important consequence of the stratification is that in order to achieve comfortable temperatures in the occupied zone near the floor of the space, temperatures become elevated higher in the space; with mixing ventilation this increases the heat loss associated with the ventilation in comparison to that in a well-mixed space, in which the temperature is maintained at the comfort temperature throughout. Using our model, we compare the cost of such stratification in terms of elevated heating load, accounting for the role of the thermal mass in controlling the strength of the stratification.

12:01PM H24.00008 Turbulent buoyant convection from a maintained source of buoyancy in a narrow vertical tank , DAAN D.J.A. VAN SOMMEREN, C.P. CAULFIELD, University of Cambridge - BP Institute/ DAMTP, ANDREW W. WOODS, University of Cambridge - BP Institute — We describe new experiments to examine the buoyancy-induced mixing which results from the injection of a small constant volume flux of fluid of density ρ_s at the top of a long narrow vertical tank with square cross-section that is filled with fluid of density $\rho_0 < \rho_s$. The injected fluid vigorously mixes with the less dense fluid that initially occupies the tank, such that a dense mixed region of turbulent fluid propagates downwards. The density at any height within this mixed region increases with time. For an ideal point source of constant buoyancy flux B_s, we show that the height of the mixed region grows as $h \sim B_s^{1/6} d^{1/3} t^{1/2}$ and that the reduced gravity $g' = g(\rho - \rho_0)/\rho_0$ at the top of the tank increases as $g'(0) \sim B_s^{5/6} d^{-7/3} t^{1/2}$, with d the width of the tank. Once the mixed region reaches the bottom of the tank, the turbulent mixing continues, and we demonstrate that the reduced gravity at each height increases approximately linearly with time. Our results are consistent with Prandtl's mixing length theory, which suggest that the local turbulent flux is given by $J = \lambda d^2 (\partial \overline{g'}/\partial z)^{3/2}$, with λ an $O(1)$ constant. We solve the corresponding nonlinear turbulent diffusion equation, and show a good agreement with experimental profiles obtained with a dye attenuation technique.

12:14PM H24.00009 Flushing of a dense fluid from an urban canyon part 1: Steady state measurements , NIGEL KAYE, ZAHRA BARATIAN, Clemson University — We consider the role of buoyancy on the vertical transport of a dense gas due to a horizontal wind flow above a street canyon. The density of the pollutant suppresses vertical mixing as the turbulent shear flow at the top of the canyon must do work to raise the dense gas up above the canyon top. We present results of a series of experiments to measure the rate of removal of a dense miscible fluid from a two dimensional square canyon open at the top. The cavity is formed by square blocks up- and down-stream. Dense fluid is introduced at a constant rate at the base of the cavity and is removed by mixing with the flow passing over the top of the cavity. Two different steady flows are observed. For higher Richardson numbers, a two layer stratification develops in which there is a relatively sharp interface. In this case the mixing is parameterized in terms of an entrainment velocity across the interface that is a function of the Richardson number and the fractional depth of the interface below the cavity top. For lower Richardson numbers no interface is observed and the buoyancy increases linearly with height above the cavity base. We also found a range of Richardson numbers for which both steady stratifications are possible and for which the steady flow depends on the initial conditions.

12:27PM H24.00010 Flushing of a dense fluid from an urban canyon part 2: Transient flows , ZAHRA BARATIAN, NIGEL KAYE, Clemson University — We consider the wind driven flushing of a finite volume of dense fluid from a canyon. Unlike the case of a continuous release of dense fluid into the canyon, no steady state flow is established. Instead the total buoyancy in the canyon decreases monotonically with time until it is completely flushed. The time taken to flush an initially full canyon depends on the initial flow Richardson number. Two flow phenomena were observed. For high Richardson numbers a two layer stratification is maintained for much of the flushing process. Dense fluid is skimmed off the top of the dense layer. However, the buoyancy of the layer also slowly decreases over time indicating that fresh ambient fluid is mixed down into the dense layer. The two layer transient flushing is modeled using an entrainment coefficient for mixing across the interface and a skimming coefficient to describe change in interface height over time. Both these coefficients were derived from steady-state constant buoyancy flux experiments. For lower Richardson numbers the fluid in the cavity is relatively well mixed and the buoyancy decays approximately exponentially over time consistent with previous observations of flushing of a non-buoyant fluid from a canyon. The exponential decay rate decreases with increasing Richardson number.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H25 Particle-laden Flows III 328

10:30AM H25.00001 Particle Impaction for Trace Contaminant Detection in a Very-High-Volume Airstream , MATTHEW FULGHUM, MICHAEL HARGATHER, GARY SETTLES, Pennsylvania State University — An impactor with automated thermal desorption has been designed and tested for airborne trace contaminant detection in a very- high-volume airstream, up to one cubic meter per second. This airflow is scanned for contaminants using an ion mobility spectrometer, which is flow rate-limited to $\sim 10\frac{cc}{s}$. As the chemicals of interest, once impacted, are capable of being thermally desorbed and interrogated as a vapor, a novel method of collecting these particles was devised. A 0.3m long linear-slot impactor is used to concentrate particles upon a periodically-heated notch of equal length in a rotating rod. After impaction, this rod is rotated into the interrogation position and is heated to thermally desorb the impacted particles into the carrier-gas flow as a vapor. This flow is several orders of magnitude smaller than the primary airstream being sampled and carries a heavy concentration of the desorbed vapor from the impacted particles to subsequent chemical analysis by the IMS, improving contaminant detection without overwhelming its flow rate capabilities.

10:43AM H25.00002 Significance of Neglected Hydrodynamic Forces on the Motion of Submerged Particles Acted on by External Body Forces , DANIEL COOPER, Virginia Tech - Wake Forest School of Biomedical Engineering & Science, JOHN CHARONKO, PAVLOS VLACHOS, Virginia Tech Department of Mechanical Engineering — Recently, the manipulation of submerged particles using electromagnetic body forces has drawn increasing interest from a variety of fields, particularly medicine, where electrophoretic manipulation in lab-on-a-chip applications and magnetic drug targeting have become important areas of interest. As a direct result of this increasing interest a large number of simulations have been performed investigating the performance of devices and systems whose operation is based upon these physics. In the vast majority of cases, these simulations are based upon a force balance of the applicable body force and Stokes drag. Such simulations neglect additional hydrodynamic forces, including the added mass, Basset, Saffman, and Magnus forces. In the current study, the full equations of motion containing all of the aforementioned terms are nondimensionalized leading to a set of nondimensional parameters governing the behavior of the particle. A parametric investigation is then performed by calculating particle trajectories for both Poiseuille and Womersley flows. This analysis reveals that in many cases, the forces neglected in previous simulations are significant and should not be neglected in future studies.

10:56AM H25.00003 Migration of suspended particles and topological defects within a nematic liquid crystal in anisotropic microfluidic devices , NICHOLAS FLOWER, RAGHAVENDRA DEVENDRA, Chemical and Biomolecular Engineering, Johns Hopkins University, Baltimore MD, JOEL ROVNER, ROBERT LEHENY, Physics and Astronomy, Johns Hopkins University, Baltimore MD, GERMAN DRAZER, Chemical and Biomolecular Engineering, Johns Hopkins University, Baltimore MD — Devendra et al. (2010) have explored the mobility of particles in periodic anisotropic arrays of obstacles. They observed that the particles migrate away from the applied force at angles that depend on the size of the particle, suggesting a method for microfluidic separation. Rovner et al. (2010) have shown that a lift force redirects particles in a nematic liquid crystal at an angle to the applied force that depends on the anchoring of the nematic director at the surface of the particle. Combining these approaches for separation of particles, we have investigated the behavior of the nematic 4-cyano-4'-pentylbiphenyl (5CB), both with and without suspended particles, flowing inside microfluidic devices containing arrays of obstacles. We report a certain degree of control over generation of topological defects in the nematic order, which we image using polarized microscopy, and control of the flow direction of these defects and suspended particles. The results suggest new ways to conduct microfluidic separation, as well as a new, controlled method for investigating the behavior of liquid crystal defects in flow.

11:09AM H25.00004 Modeling Coupled Particle/Fluid/Thermal/Charge/Ion Transport with a Hybrid Lattice Boltzmann and Immersed Boundary Method , MIAO WANG, PhD student, YING SUN, Assistant Professor — A hybrid model based on lattice Boltzmann method and immersed boundary method is developed to simulate complex transport processes in semi-solid flow batteries, where electrode slurries are composite of both electrolyte and electrode particles flowing through an electrochemical reaction zone. In such a case, the coupled charge and ion transport need to be solved in a moving frame of reference, accounting for the motion of electrode particles. The model is first validated by studying the sedimentation of non-isothermal particles in a fluid domain, where the hydrodynamic forces are determined by the lattice Boltzmann equation and the immersed boundary method is used to solve for heat transfer. The hybrid model is then applied to solving for the species and ion transport in both the electrolyte and electrode particles inside semi-solid electrode slurries. The charge/ion transport properties as a function of particle volume fraction and particle size of the electrode material are presented.

11:22AM H25.00005 3D description of particle detachment from a smooth surface[1] , ALEX LIBERZON, HADAR TRAUGOTT[2], Tel Aviv University, INTERNATIONAL COLLABORATION FOR TURBULENCE RESEARCH COLLABORATION — This study explores the necessary turbulent flow conditions for initial entrainment of particles from smooth beds into zero-mean-shear flow in an oscillating grid chamber as compared to the bounded shear flow created in a lid-driven cavity. The experiments are not designed to fully mimic the real problem of sediment transport but rather identify key mechanisms, utilizing direct observation and quantification of particle motion at the beginning, during and after lift-off. In both turbulent flows, particle image velocimetry (PIV) and three-dimensional particle tracking velocimetry (3D-PTV) are used to determine the properties of turbulent flows and to track the movement of individual particles through the various phases of the resuspension. The combination of the experimental methods allow to correlate in a quantitative manner the flow conditions responsible for rolling, pick-up, detachment and re-entrainment of particles.

[1]The study is supported by the Israel Science Foundation under grant 782/08
[2]Graduate student

11:35AM H25.00006 ABSTRACT WITHDRAWN –

11:48AM H25.00007 Capillary breakup of discontinuously shear thickening suspensions , PAWEL ZIMOCH, GARETH MCKINLEY, ANETTE HOSOI, Massachusetts Institute of Technology — Extensional rheology of discontinuously shear thickening suspensions is not well understood, in part due to unresolved experimental challenges. Such suspensions tend to sag, drain or fracture when tested using traditional tools such as filament-stretching or capillary breakup extensional rheometers. We present an alternative method of conducting capillary breakup experiments with thickening suspensions by placing them between two layers of immiscible oil. In experiments conducted with silica and cornstarch particles we observe the formation of bead-on-a-string morphologies with multiple satellite and sub-satellite bead generations, similar to the morphologies observed in breakup of viscoelastic fluids. Using a one dimensional numerical model, we show that formation of beads is a consequence of the discontinuous nature of thickening in the suspensions. Finally, we delineate the parameter regimes where formation of beads occurs, and where it is suppressed.

12:01PM H25.00008 An experimental study of gravity-driven thin-film flow with buoyant particles , WYLIE ROSENTHAL, Harvey Mudd College, PAUL LATTERMAN, SPENCER HILL, University of California, Los Angeles, PAUL DAVID, University of Southern California, MATTHEW MATA, ALIKI MAVROMOUSTAKI, ANDREA BERTOZZI, University of California, Los Angeles — Our experimental study involves silicone oil with buoyant foamed glass spheres, flowing under the action of gravity. We perform an extensive parametric study varying the angle of inclination, particle size, density and concentration. In the case of heavy particles, three regimes arise involving settling of particles to the substrate versus settling to the front of the flow. In contrast, only one regime is observed with buoyant particles, however the dynamics depart significantly from that of a clear fluid. We discuss results for front position versus time as well as changes in the fingering instability as a function of experimental parameters.

12:14PM H25.00009 Theoretical challenges in modeling gravity-driven thin-film flow with buoyant particles , PAUL DAVID, University of Southern California, SPENCER HILL, PAUL LATTERMAN, University of California, Los Angeles, WYLIE ROSENTHAL, Harvey Mudd College, ALIKI MAVROMOUSTAKI, MATTHEW MATA, ANDREA BERTOZZI, University of California, Los Angeles — We discuss the role of shear-induced migration in particle-laden thin-film flow. For heavy particles, recent experimental results show that this effect explains separation of behavior into three different regimes. In contrast only one regime is present in the case of buoyant particles although the behavior can depart significantly from that of a clear fluid. We discuss current modeling challenges for buoyant particle-laden flow including stability and dynamics of the front in the context of several available models for this problem.

12:27PM H25.00010 Bridging the Rheology of Dense Granular Flows in Three Regimes , SEBASTIAN CHIALVO, JIN SUN[1], SANKARAN SUNDARESAN, Princeton University — Using the discrete element method, simulations of simple shear flow of dense assemblies of soft, frictional particles have been carried out over a range of shear rates and volume fractions in order to characterize the rheology of granular flows in three regimes. In agreement with previous results for frictionless spheres [1], the pressure in each regime is found to obey an asymptotic power law relation with shear rate. These relations are then used to construct a blended pressure model. Additionally, we constitute the shear stress ratio in terms of two dimensionless groups: the inertia number [2], which governs the rheology of hard particles, and the ratio of shear time to the particle binary collision time, which characterizes the departure from hard-sphere behavior. The pressure and shear stress ratio relations form a rheological model that, in the hard-sphere limit, can be written as a modified kinetic theory for dense granular flows [3].

[1] T. Hatano, et al., J. Phys. Soc. Japan 76, 023001 (2007).
[2] da Cruz, F. et al., Phys. Rev. E 72, 021309 (2005).
[3] J. Jenkins, and D. Berzi, Granular Matter 12, 151 (2010).

[1]Now at the University of Edinburgh, UK

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H26 Focus Session: PIV Uncertainty 329

10:30AM H26.00001 Uncertainty of Spatial and Temporal velocity for High-speed PIV , KOJI OKAMOTO, the University of Tokyo — High-speed PIV system can provide huge amount of images for PIV analysis. With frame straddling technique, the dynamic range of velocity could be improved with four image analysis technique. This technique sacrifices the temporal directional resolution to improve the dynamic ranges. Because total amount of the image information is constant, if we improve a spatial resolution, the temporal resolutions will decrease. There should be a sufficient conversion of spatial and temporal resolutions. The relation between sampling frequency and flow characteristic frequency in both spatial and temporal direction will be discussed.

10:43AM H26.00002 Random uncertainty estimates of PIV measurements using correlation statistics , STEVE WERELEY, Purdue University — In recent years researchers have made efforts to extract scalar and even tensor quantities from the characteristics of the correlation peak. These have included temperature (Wereley, 2002, 2010), velocity distribution (Wereley 2006, Westerweel ~2006), particle hydrodynamic size (Wereley 2008) and even Reynolds stresses (Kaehler, ~2008). It is also possible to extract random uncertainty estimates from the correlation peak characteristics—provided enough is known about the flow. Obtaining uncertainty estimates in this way would provide local estimates of uncertainty rather than the current global rules of thumb that are relied upon.

10:56AM H26.00003 Estimation of Uncertainty Bounds for Individual PIV Measurements , JOHN CHARONKO, PAVLOS VLACHOS, Virginia Tech — Numerous studies have established firmly that particle image velocimetry (PIV) is a robust method for non-invasive, quantitative measurements of fluid velocity, and that when carefully conducted, typical measurements can accurately detect displacements in digital images to within a tenth of a pixel or less. However, previously, these estimates have only been able to provide guidance on the expected error for an average measurement under specific image quality and flow conditions. This work demonstrates a new method for estimating the uncertainty bounds to within a given confidence interval for a specific, individual measurement. We show that for a phase-only, generalized cross-correlation the ratio of primary to secondary peak heights correlated strongly with the range of observed error values for each individual displacement measurement, regardless of flow condition or image quality. Using an analytical model of the relationship derived from synthetic data sets, the uncertainty bounds at a 95% confidence interval are then computed for several artificial and experimental flow fields, and the true errors are shown to match the predicted uncertainties. While this method is not able to predict the true error for a given measurement, knowledge of the uncertainty level for a PIV experiment will provide great benefits in engineering design studies and CFD validation efforts.

11:09AM H26.00004 Resolution limit of Digital Particle Image and Particle Tracking Velocimetry , CHRISTIAN J. KAEHLER, SVEN SCHARNOWSKI, CHRISTIAN CIERPKA, Universitaet der Bundeswehr Muenchen — This work analyzes the spatial resolution that can be achieved by DPIV in dependency on the tracer particles and the imaging and recording system. While the in-plane resolution of DPIV is given by the interrogation window size, ensemble-correlation seams to increase the achievable resolution up to a single pixel. However, it is shown that the resolution limit of single-pixel ensemble-correlation is determined by the particle image size which is dependent on the diameter of the particles, the magnification, the f–number of the imaging system, and optical aberrations. As the minimum detectable particle image size is given by the pixel dimension of the camera sensor in DPIV, this quantity is also important for a systematic analysis of the resolution limit. It is shown that the optimal magnification that results in the best possible spatial resolution can be estimated from the particle size, the lens properties, and the pixel size of the camera. Thus, the provided information in this paper allows to optimize the camera and objective lens choices as well as the working distance for a given setup. Furthermore, it is shown that the resolution limit of DPIV can be resolved by using PTV evaluation techniques.

11:22AM H26.00005 Characteristics of seeding particles for PIV/PTV analysis , TAL HADAD, Ben-gurion University, ALEXANDER LIBERZON, Tel-Aviv University, ANNE BERNHAIM, ROI GURKA, Ben-Gurion University — PIV and PTV are non-intrusive state-of-the-art techniques widely used for flow measurements. Seeding particles are required to be used as tracers to the flow. The accuracy of the velocity measurements is limited by the ability of the tracer particles to adequately follow the instantaneous motion of the continuous phase. In order to follow the flow effectively, the particles should satisfy numerous requirements: size, sphericity, density, high refractive index, concentration and chemical inert. Since seeding particles for liquids are commonly polymer-based particles we probe the influence of their surface coating on the results obtained from optical measurements. Using a canonical lid-driven cavity flow we measured the velocity field using PIV and PTV and compared the results (velocity and acceleration) obtained with the same particles with and without chemical treatment of surfactants. Probability density functions of the results using particles before and after treatment are compared statistically utilizing the two-sample Kolmogorov-Smirnov tests. Although the mean values exhibit similar trends, fluctuations and velocity derivatives show some discrepancy in respect to the chemical treatment. The obtained results show a variance of up to 5% between the values obtained for using washed and un-washed particles, for both PIV and PTV experiments with some influence related to the size of the particles.

11:35AM H26.00006 Practical estimation of DPIV uncertainty using pseudo-image pairs[1] , MICHAEL MCPHAIL, MATTHEW WELDON, MICHAEL KRANE, ARNOLD FONTAINE, HOWARD PETRIE, ARL Penn State, JOHN BUCHANAN, Bettis Atomic Power Laboratory, DONALD LORENTZ, Betttis Atomic Power Laboratory, RICHARD BAUER, Bettis Atomic Power Laboratory — While the sources of uncertainty in PIV measurement have long been understood, quantifying the contribution of each source has proved elusive. Here we present a straightforward approach using image pairs formed from a real image and that same image, subject to a uniform non-integer displacement. This procedure thus naturally incorporates contributions of image quality, optical setup, and pixel resolution to uncertainty. The pseudo-image pairs for a range of displacements are used to estimate an ensemble of displacement vectors, which is then used to estimate the bias and random errors in displacement. This method is particularly useful for identifying pixel locking bias. Applications of this approach to both fully-developed turbulent pipe flow and impinging jet flow will be presented.

[1]Acknowledge partial support of NIH 5R01DC005642.

11:48AM H26.00007 Uncertainty in Velocity Fluctuations for Two-Component PIV Measurements[1] , BRANDON WILSON, JEFF HARRIS, BARTON SMITH, Utah State University Mechanical and Aerospace Engineering — The subpixel displacement estimator in PIV generates random fluctuations, in the velocity measurement. This noise causes velocity fluctuations to be overestimated. We quantify this overestimate of the fluctuation level due to four error sources: particle displacement, particle image size, particle density and flow gradients. A jet experiment that is designed to segregate error sources and demonstrate their effects by comparison of PIV and hot wire measurements is used. While the influence of the error sources studied on the mean velocity is negligible, increases in velocity fluctuation levels are observed for all error sources when compared to hot wire results, particularly particle displacement and flow gradients. Uncertainties in the velocity fluctuation levels are one-directional because the error sources considered can only increase the measured fluctuations. Sources that attenuate the fluctuation level (i.e. particle lag or volumetric averaging) are not studied. The amount by which the velocity fluctuation level is overestimated is consistent with previous theoretical studies and with recent studies using synthetic images.

[1]We acknowledge the support of US DOE through the INL LDRD Project NE-156.

12:01PM H26.00008 Evaluation of two-component PIV uncertainty for flow in porous media[1] , VISHAL PATIL, JAMES LIBURDY, Oregon State University — The measurement of flow in porous media is challenging due to accessibility and large range of flow passage scales. The use of PIV requires matching of refractive indices of fluid and solid phases. Slight mismatches are shown to cause significant tracking errors. In gaining PIV data at discrete planar locations along the optical axis, variations occur in the imaging magnification, and if not taken into consideration may lead to increased error. This paper addresses three forms of error in PIV measurements as they pertain to porous media flow: tracking error, bias error due to displacement gradients and perspective error. As applied to a porous bed of spherical beads, the local magnification is evaluated based on measured variations of detected bead diameter along the optical axis. The variation of magnification through the bed is then used to evaluate perspective error. The error due to displacement gradients was evaluated from correlation peak width. The bias error was also evaluated by reducing the interrogation window size and estimating the RMS difference between the two velocity estimates. The bias error evaluated using these two methods compared well. Results are shown for a flow at a pore Reynolds number of 4. Perspective errors are shown to be most significant, with total errors up to 6%.

[1]This study was supported by NSF through grant 0933857

12:14PM H26.00009 PIV estimates of dissipation: their accuracy and uncertainty , EDWIN COWEN, Cornell University — Particle image velocimetry (PIV) allows the determination of the rate of dissipation of turbulent kinetic energy, ϵ, over a two-dimensional region in space. As ϵ ranges over orders-of-magnitude its uncertainty is relatively high and we often consider a factor of 2 to be acceptable. Researchers have approached the determination of ϵ in many ways, including spectra-based estimates, structure function-based estimates, and the direct calculation from the available fluctuating gradients. In this presentation the accuracy and uncertainty of each of these approaches is reviewed and optimal methods for the accurate determination of ϵ with the narrowest uncertainty bounds are presented.

12:27PM H26.00010 When RANS Goes Wrong: Using PIV to Assess Ergodicity , W. ETHAN EAGLE, University of Michigan — Ergodic flow, where the spatial, temporal and ensemble averages, $\bar{u}_V$, $\bar{u}_T$, $\bar{u}_N$ are taken to be equal, is a widely used assumption in turbulent flows and is useful in applications from Taylor's frozen turbulence approximation to the Reynolds decomposition $u = \bar{u} +$ u′. Interestingly, ergodicity is only routinely (albeit tediously) verified by CFD grid convergence studies, while experimental assessments of ergodicity that require different spatial averages $\bar{u}_V$ (an impossibility for a given probe dimension), are not undertaken. In many cases, the unverified existence of experimental ergodicity results in spurious comparisons between computed and measured mean quantities. To investigate flow ergodicity, mean flow quantities were computed from 5 sets of 100 Stereo-PIV images of a supersonic turbulent boundary layer. To evaluate a pseudo-temporal ergodicity, $\bar{u}_t$, each set was recorded at a different laser pulse timing, d_t={500ns-900ns} Spatial ergodicity, $\bar{u}_V$, was assessed using 4 interrogation window cell sizes {2^nx2^n n=3,4,5,6.} Results suggest measuring, verifying, and reporting ergodicity enhances understanding of PIV optimization and uncertainty quantification programs and is essential when PIV experiments are compared to RANS (or LES) in validation and verification exercises.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H27 Focus Session: Wind Energy Fluid Dynamics IV Ballroom I

10:30AM H27.00001 Induction factor optimization through variable lift control , JOHN COONEY, THOMAS CORKE, ROBERT NELSON, THEODORE WILLIAMS, University of Notre Dame — Due to practical design limitations coupled with the detrimental effects posed by complex wind regimes, modern wind turbines struggle to maintain or even reach ideal operational states. With additional gains through traditional approaches becoming more difficult and costly, active lift control represents a more attractive option for future designs. Here, plasma actuators have been explored experimentally in trailing edge applications for use in attached flow regimes. This authority would be used to drive the axial induction factor toward the ideal given by the Betz limit through distributed lift control thereby enhancing energy capture. Predictions of power improvement achievable by this methodology are made with blade - element momentum theory but will eventually be demonstrated in the field at the Laboratory for Enhanced Wind Energy Design, currently under construction at the University of Notre Dame.

10:43AM H27.00002 ABSTRACT WITHDRAWN –

10:56AM H27.00003 Influence of transition on steady and unsteady wind-turbine airfoil aerodynamics[1] , ERIC PATERSON, ADAM LAVELY, GANESH VIJAYAKUMAR, JAMES BRASSEUR, The Pennsylvania State University — Laminar–flow airfoils for large stall–regulated horizontal–axis wind turbines are designed to achieve a restrained maximum lift coefficient and a broad laminar low- drag bucket under steady flow conditions and at specific Reynolds numbers. Blind- comparisons of the 2000 NREL Unsteady Aerodynamics Experiment showed large discrepancies and illustrated the need for improved physics modeling. We have studied the S809 airfoil under static and dynamic (ramp-up, ramp-down, and oscillatory) conditions, using the four-equation transition model of Langtry and Menter (2009), which has been implemented as a library accessible by an OpenFOAM RANS solver. Model validation is performed using surface–pressure and lift/drag data from U. Glasgow (2009) and OSU (1995) wind tunnel experiments. Performance of the transition model is assessed by analyzing integrated performance metrics, as well as detailed surface pressure and pressure gradient, wall–shear stress, and boundary–layer profiles and separation points. Demonstration of model performance in the light– and deep–stall regimes of dynamic stall is an important step in reducing uncertainties in full 3D simulations of turbines operating in the atmospheric boundary layer.

[1]Supported by NSF Grant 0933647

11:09AM H27.00004 Optimization of Airfoil Design for Flow Control with Plasma Actuators , THEODORE WILLIAMS, THOMAS CORKE, JOHN COONEY, University of Notre Dame — Using computer simulations and design optimization methods, this research examines the implementation of active flow control devices on wind turbine blades. Through modifications to blade geometry in order to maximize the effectiveness of flow control devices, increases in aerodynamic performance and control of aerodynamic performance are expected. Due to this compliant flow, an increase in the power output of wind turbines is able to be realized with minimal modification and investment to existing turbine blades. This is achieved through dynamic lift control via virtual camber control. Methods using strategic flow separation near the trailing edge are analyzed to obtain desired aerodynamic performance. FLUENT is used to determine the aerodynamic performance of potential turbine blade design, and the post-processing uses optimization techniques to determine an optimal blade geometry and plasma actuator operating parameters. This work motivates the research and development of novel blade designs with flow control devices that will be tested at Notre Dame's Laboratory for Enhanced Wind Energy Design.

11:22AM H27.00005 Harvesting energy via fluttering piezoelectric beams in viscous flow , DENIZ TOLGA AKCABAY, YIN LU YOUNG, University of Michigan — This work explores the idea of harvesting energy from ambient flows using flexible piezoelectric beams. Beams lose their stability and flutter above a critical length or flow speed or below a critical stiffness. During flutter, beams oscillate in increasing amplitude until they enter a self-sustained limit cycle oscillation, which could be exploited to harvest energy. The objectives of this study are to: (i) identify the flutter boundary of a flexible beam in viscous flow; (ii) explore the energy harvesting potential; and (iii) identify critical non-dimensional parameters and parametric relations that govern the response and stability of thin composite beams vibrating in a viscous fluid. Two-dimensional Navier-Stokes equations are solved with a nonlinear beam model coupled with a linear piezoelectric material constitutive model. The harvested energy potential for various solid/fluid combinations is investigated by varying the critical non-dimensional parameters, which are defined in terms of beam length, density, thickness, and stiffness; fluid speed and density; and piezoelectric material properties.

11:35AM H27.00006 Improving the Efficiency of Piezoelectric Fluidic Energy Harvesters[1] , CHLOE DUQUESNOIS, INSA de Lyon, France, HUSEYIN DOGUS AKAYDIN, NIELL ELVIN, YIANNIS ANDREOPOULOS, The City College of New York, USA — Piezoelectric systems can harvest electrical energy from fluid flows. Maximizing the power output and increasing their overall energy conversion efficiency is a formidable challenge. We compare the conversion efficiencies of various harvester configurations. The first experimentally tested configuration is a cantilevered beam in the wake of a circular cylinder. An efficiency of only 0.03 per cent in converting fluidic to mechanical energy (i.e. "aeroelastic efficiency") was estimated while the efficiency of mechanical to electrical energy conversion (i.e. "electromechanical efficiency") was close to 11 per cent. In our latest experiments, we attached a circular cylinder to the free end of a piezoelectric beam. The aeroelastic efficiency increased to 2.5 per cent and the electromechanical efficiency is increased to 30 per cent. Placing a stationary circular cylinder upstream of this configuration caused a dramatic increase in harvested power and efficiency. Furthermore, attaching a cylinder of a half-circle cross section resulted in a much larger power and efficiency as compared to attaching a circular cylinder to the tip of beam.

[1]Sponsored by NSF Grant: CBET #1033117.

11:48AM H27.00007 ABSTRACT WITHDRAWN –

12:01PM H27.00008 Preliminary Investigation of the Active Flow Control Benefits on Wind Turbine Blades[1] , GUANNAN WANG, JAKUB WALCZAK, MARK GLAUSER, BASMAN ELHADIDI, Syracuse University — This study investigates the benefit of flow control over a 2D airfoil specially designed for wind turbine applications. The experiments were carried out in Syracuse University's Anechoic Wind Tunnel, both with and without large scale unsteadiness in the freestream. When there is no large scale unsteadiness introduced in the flow, under open loop flow control conditions with unsteady blowing, the leading edge separation was delayed and maximum lift coefficient was increased. For the cases where large scale unsteadiness was introduced into the flow, the experiments showed that both open loop and closed loop control cases were able to reduce lift fluctuations by a measurable amount. However, only the closed loop control case which utilized surface pressure information from the airfoil near the leading edge was capable of consistently mitigating the fluctuating load.

[1]The authors gratefully acknowledge the support from the U.S Department of Energy through University of Minnesota Wind Energy Consortium, Syracuse University and the Syracuse Center of Excellence

12:14PM H27.00009 ABSTRACT WITHDRAWN —

12:27PM H27.00010 Energy-harvesting potential of multiple elastic structures in tandem arrangement

, BO YIN, HAOXIANG LUO, Vanderbilt University — Vortex-induced flapping vibrations of elastic structures attached with piezoelectric materials, i.e., "piezo-leaves", have recently been explored for its potential application in wind energy harvesting (e.g., Li, Yuan, and Lipson, J. Appl. Phys., 2011). In this work, we explore the possibility of enhancing the structural vibration and energy harvesting performance of the generator by putting the leaves in tandem arrangement and within close range of hydrodynamic interaction. A two-dimensional model is developed, where two or more elastic plates are mounted in a cross flow. In the case of two plates, the numerical simulation shows that at a particular distance, the vibration of the downstream plate is greatly increased, and so is the energy level of the entire system. For multiple plates, we observed both synchronized and apparently chaotic vibration modes. The characteristics of the vortex interaction, plate deformation, and energetics will be presented for those coupling modes.

Monday, November 21, 2011 10:30AM - 12:40PM —

Session H28 Hydrodynamic Locomotion Ballroom II

10:30AM H28.00001 Hydrodynamics of self-propulsion near boundaries

, SAVERIO SPAGNOLIE, ERIC LAUGA, University of California, San Diego — The swimming kinematics and trajectories of many microorganisms are altered by the presence of nearby boundaries, be they solid or deformable, and often in perplexing fashion. When an organism's swimming dynamics vary near such boundaries a question arises naturally: is the change in behavior biological, fluid mechanical, or perhaps mediated by other physical laws? Of interest then is a general framework for determining the extent to which fluid mechanics passively alter swimming behaviors in the presence of a wall. To this end, we explore a far-field description of swimming organisms and provide a general framework for studying the fluid-mediated modifications to swimming trajectories, and consider trapped/escape trajectories and equilibria for model organisms of varying shape and propulsive activity. This framework may help to explain surprising behaviors observed in the swimming of many microorganisms and synthetic swimming devices.

10:43AM H28.00002 Benefits of unsteady swimming near a wall

, DANIEL QUINN, PETER DEWEY, KEITH MOORED, ALEXANDER SMITS, Princeton University — The benefits of flying and swimming near the ground have been well-documented for fixed-wing vehicles, and have led to 'wing-in-ground' craft (WIG) with higher efficiencies than their conventional counterparts. Here it is examined whether unsteady propulsion techniques experience these same enhancements. Experimental particle image velocimetry was conducted in the wake of a rigid pitching panel and a flexible triangular fin, both actuated at several distances from a fixed wall. In both cases, an increase in momentum flux behind the trailing edge was observed, suggesting thrust amplification is present. A finite core vortex array model was developed to model the wake behind these propulsors. Mirror image vortex cores were placed across the wall to satisfy the zero flux boundary condition, and the effects of viscosity were estimated by assuming a slowly expanding Gaussian distribution of vorticity around each core. The model offers insight into the origins of the momentum amplification due to the presence of the wall. Supported by ONR MURI Grant N00014-08-1-0642.

10:56AM H28.00003 Wake interactions of panels swimming in a side-by-side configuration[1]

, BIRGITT BOSCHITSCH, PETER A. DEWEY, KEITH W. MOORED, ALEXANDER J. SMITS, Princeton University — A pair of pitching panels arranged in a side-by-side configuration are experimentally examined under free swimming conditions. The panels were pitched about their leading edges by shafts located just behind the trailing edge of a NACA 0012-64 airfoil to suppress the formation of leading edge vortices. A recirculating water channel is set with a flow speed that matches the free swimming speed of the panel system. Power measurements are used to determine the energy consumption of the panel system per distance traveled. Finite and infinite aspect ratio panels are examined for a range of Strouhal numbers, transverse panel spacings, and oscillation phase shifts between the two panels. It is found that under certain operating conditions, a pair of panels maintain a higher free swimming velocity in comparison to a single isolated panel. To assess the wake generated by the panels, digital particle image velocimetry (DPIV) and hydrogen bubble visualization are used.

[1]Supported by ONR MURI Grant N00014-08-1-0642.

11:09AM H28.00004 Interaction of in-phase and out-of-phase flexible filament in fish schooling[1]

, EMAD UD DIN, HYUNG JIN SUNG, KAIST — Fish schooling is not merely a social behavior; schooling improves the efficiency of movement within the fluid environment. Inspired by the schooling from a hydrodynamic perspective, a group of aquatic animals is modeled as a collection of individuals arranged in a combination of tandem and side-by-side (diamond) formation. The downstream bodies are strongly influenced by the vortices shed by the upstream body shown by vortex-vortex and vortex-body interactions. Trailing fish takes advantage of this flow pattern for energy economy. To investigate the interactions between flexible bodies and vortices, in the present study three flexible flags in viscous flow are solved by numerical simulation using an improved version of the immersed boundary method for in-phase and out-of-phase filaments. The drag coefficient of the downstream filaments drops even below the value of a single flag. Such drag variations are influenced by the interactions between vortices shed by the upstream flexible body and vortices surrounding the downstream filaments. Interaction of the flexible flags is investigated as a function of the gap distance between flags and different bending coefficients, for in-phase and out-of-phase cases at intermediate Reynolds numbers.

[1]This study was supported by the Creative Research Initiatives of NRF/MEST (No. 2011-0000423) of Korea

11:22AM H28.00005 Turns and maneuvers during swimming[1]

, AMNEET BHALLA, NOAH MOSBERG, RAHUL BALE, NEELESH PATANKAR, Department of Mechanical Engineering, Northwestern University — In this work we use fully resolved fluid dynamics computations based on an immersed body approach to study fish turns and maneuvers. We present a numerical method to control the trajectory of fish during turns and maneuvers. Fish tracking a prey is presented as an example case. Numerical simulations are carried out on spatially adaptive grid for speed and accuracy. The effect of deformation kinematics and Reynolds number (Re), on the turn radius of an undulatory swimmer, is studied. Power spent during turning at different turn radii and Re is also reported. These results can be used to quantify the cost of various maneuvers and to identify efficient maneuvers to attain the same objective, e.g., reaching a target location during prey tracking.

[1]NSF support is gratefully acknowledged.

11:35AM H28.00006 Analyzing the fast-start performance of northern pike using a mechanical fish, YAHYA MODARRES-SADEGHI, CHENGCHENG FENG, BRIAN BONAFILIA, ANDREW COSTAIN, UMass Amherst — The northern pike is able to achieve an instantaneous acceleration of 245 m/s^2 through a two-stage motion. In the first stage the fish curls its body into either a C-shaped or an S-shaped curve (preparatory stage), and in the second stage uncurls it very quickly (propulsive stage) generating high accelerations due to the vortices shed from its tail. We have built a mechanical fish, based on the body profile of a pike, which is capable of performing this two-stage fast-start motion. Movement is governed by servo motors, which pull on cables attached to certain ribs, bending the fish into a C- or an S-shape. The degree of bending and timing of strokes can be controlled, and the fish can perform either a propulsive stroke only or a full stroke consisting of both the preparatory stage and the propulsive stage. The mechanical fish is capable of achieving peak accelerations of around 4 m/s^2. We use this fish in order to study the influence of various variables on the observed acceleration. Although the maximum accelerations observed in our mechanical fish are smaller than those of a live fish, the form of the measured acceleration signal as function of time is quite similar to that of a live fish. The hydrodynamic efficiencies are observed to be around 12%, and it is shown that the majority of the thrust is produced at the rear part of the mechanical fish – similarly to the live fish.

11:48AM H28.00007 Hydrodynamic optimality of ribbon fin shapes[1], RAHUL BALE, MALCOLM MACIVER, NEELESH PATANKAR, Department of Mechanical Engineering, Northwestern University — The primary mode of propulsion in gymnotiform and balistiform swimmers is via the undulation of anal and/or dorsal fins, commonly referred to as ribbon fins, attached to a more or less rigid body. Ribbon fins usually have a convex shape as opposed to a rectangular or concave profile. In this work we investigate if there is a hydrodynamic basis underlying this observation. Fully resolved fluid dynamics computations are performed to calculate the mechanical cost of transport (COT) as a measure of swimming efficiency of the fin. We find that the ribbon fin of a black ghost knifefish has lower COT compared to a hypothetical rectangular ribbon fin. In order to quantify this difference in COT between the two fin shapes, we obtain scaling for COT in terms of various parameters which affect the swimming performance of the fin. Using scaling arguments we address the question of how a convex profile, commonly observed in gymnotiform and balistiform swimmers, is optimal compared to rectangular or concave shapes.

[1]NSF support is gratefully acknowledged.

12:01PM H28.00008 Wake topology of under-actuated rajiform batoid robots, PABLO VALDIVIA Y ALVARADO, GABRIEL WEYMOUTH, DILIP THEKOODAN, Singapore-MIT Alliance for Research and Technology, NICHOLAS PATRIKALAKIS, Massachusetts Institute of Technology — Under-actuated continuous soft robots are designed to have modes of vibration that match desired body motions using minimal actuation. The desired modes of vibration are enabled by flexible continuous bodies with heterogenous material distributions. Errors or intentional approximations in the manufactured material distributions alter the achieved body motions and influence the resulting locomotion performance. An under-actuated continuous soft robot designed to mimic rajiform batoids such as stingrays is used to investigate the influence that fin kinematics variations have on wake topology, and the trade-offs that simplifying the body material structure has on achievable swimming performance. Pectoral fin kinematics in rajiform batoids are defined by traveling waves along the fin cord with particular amplitude envelopes along both the fin cord and span. Digital particle image velocimetry (DPIV) analysis of a prototype's wake structure and immersed-boundary numerical simulations are used to clarify the role of traveling wave wavelength, fin flapping frequency, and amplitude envelope characteristics on the resulting wake topology and swimming performance.

12:14PM H28.00009 Kinematics and dynamics of spheniscifom wings, FLAVIO NOCA, FABIEN CRISINEL, PIERRE MUNIER, hepia — Three-dimensional scans of three different species of taxidermied penguins (*Aptenodytes patagonicus*, *Pygoscelis papua*, and *Spheniscus magellanicus*) have been performed. A three-dimensional reproduction of an African penguin (*Sphenicus demersus*) wing was manufactured and tested in a hydrodynamic channel. A six-degree-of-freedom robot was programmed to perform the three dimensional kinematics, obtained from actual footage. A six-component force balance was used to retrieve the dynamics of the wing motion. Results will be presented and discussed.

12:27PM H28.00010 Flow Field Characteristics of Finite-span Hydrofoils with Leading Edge Protuberances[1], DERRICK CUSTODIO, Worcester Polytechnic Institute, CHARLES HENOCH, Naval Undersea Warfare Center, Newport, RI, HAMID JOHARI, California State University, Northridge, OFFICE OF NAVAL RESEARCH COLLABORATION — Past work has shown that humpback whale-like leading edge protuberances can significantly alter the load characteristics of both 2D and finite-span hydrofoils. To understand the mechanisms responsible for observed performance changes, the flow field characteristics of a baseline hydrofoil and models with leading edge protuberances were examined using the Stereo Particle Image Velocimetry (SPIV) technique. The near surface flow field on the hydrofoils was measured along with the tip vortex flow field on finite-span hydrofoils. Angles of attack ranging from 6 to 24 degrees were examined at freestream velocities of 1.8 m/s and 4.5 m/s, corresponding to Reynolds numbers of 180 and 450 thousand, respectively. While Reynolds number does not play a major role in establishing the flow field trends, both the protuberance geometry and spatial proximity to protuberances affect the velocity and vorticity characteristics near the foil surface, and in the wake and tip vortex. Near surface measurements reveal counter-rotating vortices on protuberance shoulders, while tip vortex measurements show that streamwise vorticity can be strongly affected by the presence of protuberances. The observed flow field characteristics will be presented.

[1]Sponsored by the ONR-ULI program.

Monday, November 21, 2011 10:30AM - 12:40PM –
Session H29 Biofluids: Cellular IV: Disease and Microorganisms Ballroom III

10:30AM H29.00001 Imaging Morphological Changes in Live-cells at various time-scales using Heterodyne Mach-Zehnder Interferometer, SHIJU JOSEPH, DAVID NEWPORT, YONGLI LI, University of Limerick, BERNIE WOULFE, Mid-West Regional Hospital — Measurement of the dynamic behavior of the cell will provide new insights about the state of the live cells, since this response depends upon its structure and functional state. Studies have shown that all mammalian cells exhibit continuous regional motion and shape changes. This is controlled by the dynamic cytoskeleton of the cell. Existing measurement techniques are either limited to point observation or do not have required speed and accuracy. The optical arrangement consists of a Mach-Zehnder interferometer integrated to a microscope. Heterodyning is achieved using a pair of AOMs. Temporal phase shifting technique is used to extract the continuously varying phase information, which is caused by the changes in cell. Due to dynamic phase change, the continuous wave signal reaching the detector is a frequency modulated signal. To extract dynamic phase, at first the instantaneous frequency of the phase modulated signal is determined, which is then integrated with respect to time to obtain time-varying phase. Results obtained for in vitro live 3T3 Fibroblast cells and REH Leukocyte cell lines are presented. Phase imaging of live leukemic cells, can be used for studying morphological differences between various sub-types of the cancerous cells.

10:43AM H29.00002 How malaria merozoites reduce the deformability of infected RBC , MAJID HOSSEINI, JAMES FENG[1], Department of Chemical and Biological Engineering, University of British Columbia, Vancouver, BC V6T 1Z3, Canada — This talk presents a three-dimensional particle-based model for the red blood cell (RBC), and uses it to explore the changes in the deformability of RBC due to presence of malaria parasite. The cell membrane is represented by a set of discrete particles connected by nonlinear springs that represent shear and bending elasticity. The cytoplasm and the external liquid are modeled as homogeneous Newtonian fluids, and discretized by particles as in standard smoothed-particle-hydrodynamics models. The merozoite is modeled as an aggregate of particles constrained to rigid-body motion. The fluid flow and membrane deformation are computed, via the particle motion, by a two-step explicit scheme, with model parameters determined from experiments. The stretching of healthy and infected RBC by optical tweezers has been simulated to investigate the contribution of rigid merozoites to the decrease in deformability.

[1]Department of Mathematics, University of British Columbia, Vancouver, BC V6T 1Z2, Canada

10:56AM H29.00003 Wetting dynamics of living drops , STEPHANE DOUEZAN, KARINE GUEVORKIAN, SYLVIE DUFOUR, DAMIEN CUVELIER, FRANCOISE BROCHARD-WYART, Institut Curie — Tissue spreading is a fundamental process in embryonic development, wound healing, and cancer invasion. We study the spreading dynamics of cell aggregates on solid substrates by means of an analogy with the wetting of a viscoelastic drop. At long times, a precursor film of cells spreads around the aggregate with two possible states: either a liquid state (cohesive migration) or a 2D gas state (where cells escape individually) depending on the cell-cell adhesion. These results provide insight into the progression of a non-invasive tumor into a metastatic malignant carcinoma.

11:09AM H29.00004 Gold nanoparticle incorporation in the cancer cells : imaging and treatment , SUNGSOOK AHN, SUNG YONG JUNG, EUN SEOK SEO, JEONGEUN RYU, SANG JOON LEE, POSTECH, POSTECH TEAM — Surface modified gold nanoparticles ($\sim$ 20 nm) are selectively incorporated in the various cancer cells. Depending on the attached molecules on the gold nanoparticle surface, incorporation efficiency of the gold nanoparticles in the cancer cells are differentiated. Two-photon fluorescence microscopy, energy dispersive X-ray spectroscopy (EDS) and second ion mass spectroscopy (SIMS) are utilized to quantify the gold nanoparticles incorporated in the cancer cells. Static images of the cancer cell are obtained by scanning electron microscopy (SEM) and zone-plate X-ray nanoscopy. On the other hand, dynamic flow images are captured by dynamic X-ray imaging. To enhance the selective incorporation into the cancer cells, specially designed aptamer is introduced on the gold nanoparticles, which changes the mechanisms of gold nanoparticle incorporation through the cancer cell membrane. Anti-cancer drugs are also incorporated, by which sustained drug delivery mechanisms are investigated. This study would contribute to the basic understanding on the nanoparticle- mediated disease treatment and advanced imaging technology.

11:22AM H29.00005 Effect of Magnetic Field on Ion Transport in bacterial cells , SAMINA MASOOD — We calculate the fluid parameters as a function of the constant static magnetic field. The ion transport is affected by the strength and the direction of the magnetic field. This effect can indicate unusual behavior of micro and nano systems. A possible explanation of the abnormal behavior is, therefore, investigated using quantum electrodynamics of many body systems. We have demonstrated the effect of magnetic field on fluid parameters through the transport of ions in the bacterial cells. We could experimentally observe an impact of this effect on the bacterial growth. However, it is still difficult to precisely estimate or compute this effect theoretically.

11:35AM H29.00006 Hydrodynamic Contributions to Amoeboid Cell Motility , OWEN LEWIS, ROBERT GUY, University of California - Davis — Understanding the methods by which cells move is a fundamental problem in modern biology. Recent evidence has shown that the fluid dynamics of cytoplasm can play a vital role in cellular motility. The slime mold Physarum polycephalum provides an excellent model organism for the study of amoeboid motion. In this research, we use both analytic and computational models to investigate intracellular fluid flow in a simple model of Physarum. In both models, of we are specifically interested in stresses generated by cytoplasmic flow which act in the direction of cellular motility. In our numerical model, the Immersed Boundary Method is used to account for such stresses. We investigate the relationship between contraction waves, ?ow waves and locomotive forces, and attempt characterize conditions necessary to generate directed motion.

11:48AM H29.00007 Intrinsic viscosity of actively swimming microalgae suspensions , RANDY EWOLDT, University of Illinois at Urbana-Champaign, LUCAS CARETTA, ANWAR CHENGALA, JIAN SHENG, University of Minnesota — Suspensions of actively swimming microorganisms exhibit an effective viscosity which may depend on volume fraction, cell shape, and the nature of locomotion (e.g. "pushers" vs. "pullers"). Although several dilute-regime theories have been offered for active suspensions, no experimental study to our knowledge has been able to resolve the dilute-regime intrinsic viscosity of actively swimming microorganism suspensions. Here we use a cone-and-plate rheometer to experimentally measure the dynamic shear viscosity for motile and non-motile suspensions of unicellular green algae (Dunaliella primolecta, a biflagellated "puller"). The low viscosity biological samples require careful experimental protocols to avoid settling and flow-induced migration, and to minimize precision error. With these protocols in place we can distinguish the intrinsic viscosity which we show is higher for the motile "puller" swimmers compared to the immobilized counterparts. This observation is consistent with recently proposed dilute-regime theories which predict that "pullers" should have a higher viscosity than non-motile suspensions.

12:01PM H29.00008 Hydrodynamics of cell–cell mechanical signaling , ROLAND BOUFFANAIS, Singapore University of Technology and Design, DICK K.P. YUE, Massachusetts Institute of Technology — Mechanotactic cell motility, i.e. directed motion mediated by a mechanical stress at the cell's surface, has recently been shown to be a key player in the initial aggregation of crawling cells such as leukocytes and amoebae. The effects of mechanotactic signaling in the early aggregation of amoeboid cells has been investigated using a general mathematical model based upon known biological evidence. We elucidate the hydrodynamic fundamentals of the direct guiding of a cell through mechanotaxis in the case where one cell transmits a mechanotactic signal through the fluid flow by changing its shape. It is found that any mechanosensing cells placed in the stimulus field of mechanical stress are able to determine the signal transmission direction with a certain angular dispersion which does not preclude the aggregation from happening. The ubiquitous presence of noise is accounted for by the model.

12:14PM H29.00009 Micro PIV Measurements of the Internal Flow of an *Amoeba proteus* , CHRISTIAN RESAGK, ELKA LOBUTOVA, LING LI, Ilmenau University of Technology, Institute of Thermodynamics and Fluid Mechanics, DANJA VOGES, Ilmenau University of Technology, Department of Biomechatronics — We report about micro PIV measurements of the internal flow in the protoplasm of an amoeba. The velocity data shall give information about the mechanism of the change of amoeba's contour during its locomotion in water. The experimental data is used for an analytical modeling of the locomotion mechanism with the help of a variable contour and finally for the development of locomotion principles for micro robots. The experimental set-up consists of a microscope and a CCD camera with 12 frames per second and image analysis software. The illumination of the amoeba was done by the built-in microscope halogen lamp. We use the phase contrast configuration to capture images of the amoeba moving in water. We applied an electrical field to the water channel in order to control the movement of the amoeba in one direction. During this motion we measured time dependent velocity vector fields of the protoplasm flow, estimated velocity profiles and analyzed time series of the maximum velocity. The velocity vector plots are calculated from the images by using cross correlation and naturally occurring particles in the protoplasm. Beside the analyses of the internal flow we recorded the motion of the center of gravity and the variation of the sectional area.

12:27PM H29.00010 Numerical study on the dynamics and oxygen transport of a healthy red blood cell and a malaria-infected red blood cell

PAHALA GEDARA JAYATHILAKE, Singapore-MIT Alliance, National University of Singapore, LIU GANG, KHOO BOO CHEONG, Department of Mechanical Engineering, National University of Singapore — In the present work, a red blood cell (RBC) and a malaria-infected red blood cell (IRBC) moving along a capillary are simulated with including their permeable properties of the membranes by using a numerical technique based on the two-dimensional immersed interface method. The adhesiveness of the IRBC membrane is modeled by means of a potential function. Then, the model is employed to simulate the motion of a biconcave RBC in the absence of membrane stickiness and a more rigid and circular IRBC in the presence of membrane stickiness. The results show that the RBC gradually moves away from the capillary wall while the IRBC rolls on the capillary wall due to its stickiness. This rolling behavior of the IRBC agrees well with experimental findings. It is found that the resistance on the plasma flow given by the IRBC is larger than the corresponding resistance given by the RBC revealing that macrovascular blockage could happen due to malaria infection. Furthermore, oxygen transport in capillaries and oxygen absorption by nearby muscles are investigated in the presence of a RBC and an IRBC.

Monday, November 21, 2011 10:30AM - 12:40PM –

Session H30 Biofluids: Phonation and Upper Airway Flows Ballroom IV

10:30AM H30.00001 Irregular vocal fold dynamics incited by asymmetric fluid loading in a model of recurrent laryngeal nerve paralysis

DAVID SOMMER, University of Waterloo, BYRON D. ERATH, The George Washington University, MATIAS ZANARTU, Universidad Tecnica Federico Santa Maria, SEAN D. PETERSON, University of Waterloo — Voiced speech is produced by dynamic fluid-structure interactions in the larynx. Traditionally, reduced order models of speech have relied upon simplified inviscid flow solvers to prescribe the fluid loadings that drive vocal fold motion, neglecting viscous flow effects that occur naturally in voiced speech. Viscous phenomena, such as skewing of the intraglottal jet, have the most pronounced effect on voiced speech in cases of vocal fold paralysis where one vocal fold loses some, or all, muscular control. The impact of asymmetric intraglottal flow in pathological speech is captured in a reduced order two-mass model of speech by coupling a boundary-layer estimation of the asymmetric pressures with asymmetric tissue parameters that are representative of recurrent laryngeal nerve paralysis. Nonlinear analysis identifies the emergence of irregular and chaotic vocal fold dynamics at values representative of pathological speech conditions.

10:43AM H30.00002 Fluid-acoustic interactions and their impact on pathological voiced speech[1]

BYRON D. ERATH, The George Washington University, MATIAS ZANARTU, Universidad Tecnica Federico Santa María, SEAN D. PETERSON, University of Waterloo, MICHAEL W. PLESNIAK, The George Washington University — Voiced speech is produced by vibration of the vocal fold structures. Vocal fold dynamics arise from aerodynamic pressure loadings, tissue properties, and acoustic modulation of the driving pressures. Recent speech science advancements have produced a physiologically-realistic fluid flow solver (BLEAP) capable of prescribing asymmetric intraglottal flow attachment that can be easily assimilated into reduced order models of speech. The BLEAP flow solver is extended to incorporate acoustic loading and sound propagation in the vocal tract by implementing a wave reflection analog approach for sound propagation based on the governing BLEAP equations. This enhanced physiological description of the physics of voiced speech is implemented into a two-mass model of speech. The impact of fluid-acoustic interactions on vocal fold dynamics is elucidated for both normal and pathological speech through linear and nonlinear analysis techniques.

[1]Supported by NSF Grant CBET-1036280.

10:56AM H30.00003 Dynamically Scaled Glottal Flow Through Symmetrically Oscillating Vocal Fold Models

LORI HALVORSON, University of Nebraska, ANDREW BAITINGER, Troy High School, ERICA SHERMAN, University of Nebraska, MICHAEL KRANE, Penn State ARL, LUCY ZHANG, RPI, TIMOTHY WEI, University of Nebraska — Experimental results derived from DPIV measurements in a scaled up dynamic human vocal fold model are presented. The 10x scale vocal fold model is a new design that incorporates key features of vocal fold oscillatory motion. This includes coupling of down/upstream rocking as well as the oscillatory open/close motions. Experiments were dynamically scaled to examine a range of frequencies, 100 – 200 Hz, corresponding to the male and female voice. By using water as the working fluid, very high resolution, both spatial and temporal resolution, was achieved. Time resolved movies of flow through symmetrically oscillating vocal folds will be presented. Both individual realizations as well as phase-averaged data will be shown. Key features, such as randomness and development time of the Coanda effect, vortex shedding, and volume flow rate data will be shown. In this talk, effects associated with paralysis of one vocal fold will be discussed. This talk provides the baseline fluid dynamics for the vocal fold paralysis study presented in Sherman, *et al*. Supported by the NIH.

11:09AM H30.00004 Examination of the Vocal Fold Paralysis on the Fluid Dynamics of the Glottis

ERICA SHERMAN, University of Nebraska, MICHAEL KRANE, Penn state ARL, LUCY ZHANG, RPI, TIMOTHY WEI, University of Nebraska — This talk is coupled to the symmetric vocal fold oscillation study presented in Halvorson, *et al*. In this study, one of the two symmetric vocal fold models was allowed to remain rigid while the other model was driven through a normal oscillation cycle. Again, a range of reduced frequencies were studied corresponding to physiological frequencies from 100 – 200 Hz. Flow measurements showing jet velocity and orientation, vortex shedding as a function of time through an oscillation cycle will be presented. Experimental data has been phase averaged to highlight characteristic differences between male and female voices. Additionally, volumetric flow rate and glottal behavior will be presented to show recurring features in phonation during an oscillation cycle. An example of differences between the paralysis case and the symmetrically oscillating vocal fold case is that the Coanda effect develops much more quickly and predictably for the paralysis case. Additional comparisons between diseased and healthy conditions will be presented and discussed. Supported by the NIH.

11:22AM H30.00005 Experimental study of the effects of airflow and vocal fold stiffness on male and female voice production[1]

ELIZABETH CAMPO, MCPHAIL MICHAEL, KRANE MICHAEL, ARL Penn State — The effect of airflow in voice production is not fully understood, leading to difficulties when clinically diagnosing voice disorders. Many existing studies in this this area focus primarily on the male physiology. This study incorporates 2-layer, molded silicone vocal fold models whose geometry mimics the shape and dimensions of both male and female vocal folds. Measured quantities include subglottal and transglottal pressure, volume flow rate, and radiated sound. The results are used to clarify the relationship between glottal airflow and sound production. The Implications of the measurements for similarities and differences between male and phonation are discussed.

[1]Acknowledge support of NIH grant 5R01DC005642 and ARL E&F program.

11:35AM H30.00006 Numerical study of human vocal folds vibration using Immersed Finite Element Method[1] , XINGSHI WANG, LUCY ZHANG, MICHAEL KRANE, Rensselaer Polytechnic Institute — The voice production procedure is a self-oscillating, fluid-structure interaction problem. In this study, the vocal folds vibration during phonation will be simulated by self-oscillated layered-structure vocal folds model, using Immersed Finite Element Method. With the numerical results, we will find out the vocal folds vibration pattern, and also show how the lung pressure, stiffness and geometry of vocal folds will affect the vocal folds vibration. With further analysis, we shall get better understanding of the dynamics of voice production.

[1]National Institute on Deafness and Other Communication Disorders

11:48AM H30.00007 Aeroacoustic simulation for phonation modeling[1] , JEFFREY IRWIN, AMANDA HANFORD, BRENT CRAVEN, MICHAEL KRANE, ARL Penn State — The phonation process occurs as air expelled from the lungs creates a pressure drop and a subsequent air flow across the larynx. The fluid-structure interaction between the turbulent air flow and oscillating vocal folds, combined with additional resonance in the oral and nasal cavities, creates much of what we hear in the human voice. As many voice-related disorders can be traced to irregular vocal tract shape or motion, it is important to understand in detail the physics involved in the phonation process. To numerically compute the physics of phonation, a solver must be able to accurately model acoustic airflow through a moving domain. The open-source CFD package OpenFOAM is currently being used to evaluate existing solvers against simple acoustic test cases, including an open-ended resonator and an expansion chamber, both of which utilize boundary conditions simulating acoustic sources as well as anechoic termination. Results of these test cases will be presented and compared with theory, and the future development of a three-dimensional vocal tract model and custom-mode acoustic solver will be discussed.

[1]Acknowledge support of NIH grant 5R01DC005642 and ARL E&F program.

12:01PM H30.00008 Estimation of aeroacoustic source strengths in phonation[1] , MICHAEL KRANE, ELIZABETH CAMPO, MICHAEL MCPHAIL, ARL Penn State — The underlying mechanisms of phonatory sound production are studied by quantifying the strength of the aerocoustic sources. The sources include a volume source due to vocal fold displacement, and a dipole related to the transglottal pressure difference. The challenges involved with estimating these sources in a resonator are addressed. Measurements of flow-induced vibration and sound produced by life-scale, 2-layer silicone rubber model vocal folds were conducted in a physical model of the human airway. Measurements, including transglottal pressure, radiated sound, and high-speed imaging of the model glottis, were used to produce estimates of the strengths of volume and dipole sources.

[1]Acknowledge support of NIH grant 5R01DC005642 and ARL E&F program.

12:14PM H30.00009 Epiglottal Flow Physics[1] , ANDREW POLLARD, Queen's University, ABDUL-MONSIF SHINNEEB — PIV measurements have been made at three locations in the pharynx/larynx region in the ETA model, one along the central sagittal plane and two cross-sectional planes. The measurements were made at a flow rate of 9.04 l/min which corresponds approximately to 10 l/min in the prototype. The corresponding Reynolds number Re based on the inlet condition is 716. Two thousand images were acquired at each location at a framing rate of 2 Hz. The mean velocity fields were then calculated. In addition,the data was analysed by the proper orthogonal decomposition (POD) technique to expose vortical structures. Only few modes were used for the POD reconstruction which recovered about 60% of the turbulent kinetic energy. The results showed that the flow is characterised by regions of re-circulation, jet-like, and sink-like flows. In addition, the POD-reconstructed fields revealed some interesting features that occur in the human pharynx/larynx region near the epiglottis such as tearing and pairing processes, as well as the interaction between the flows induced by the structures.

[1]Funded by NSERC

12:27PM H30.00010 Numerical simulation of transitional flow in a human upper airway segment in the presence of uncertainty , OLAF MARXEN, von Karman Institute for Fluid Dynamics — The flow in human airways may be laminar, transitional, or turbulent in different airway segments. Specifically, laminar-turbulent transition is believed to occur in the larynx or in the trachea. Present approaches to simulate such flows typically employ numerical methods solving the steady Reynolds-averaged Navier-Stokes equations. However, natural airway deformations or pathological obstructions such as tumors may generate recirculation zones and lead to highly unsteady flow features that are not well captured by these numerical methods. We perform direct numerical simulations of transitional flow through a pipe-like canonical geometry representative of an airway segment. The incompressible Navier-Stokes equations in conjunction with an immersed boundary method are solved to simulate the unsteady flow. In order to model perturbations present in the incoming flow, small-amplitude disturbances are forced to explicitly trigger flow instabilities. Time-dependent inflow profiles are applied to model the change in flow velocity during the breathing process. In order to account for natural variability during breathing, the inflow profile is treated as an uncertain function. Resulting uncertainty in the flow field is quantified using stochastic collocation.

Monday, November 21, 2011 2:00PM - 2:35PM –
Session J27 Invited Session: Fluid Mechanics and Public Policy Including Natural Cisasters and Climate Change Ballroom I-II

2:00PM J27.00001 Fluid Mechanics and Public Policy including natural disasters and climate change , JULIAN HUNT, UCL, Trinity College Cambridge, TU Delft — Where public policy relies on science for dealing with important issues affecting society, scientists involved have to explain what is known, what is uncertain, what might be known with more research and how to integrate science into decisions. Lessons have been learnt about all these stages in recent science-policy issues. Fluid mechanics and related sciences are contributing significantly to many problems. Some are specific such as the internal structure of extreme flow events in the atmosphere and ocean, or the microphysics of particle coagulation in narrow passages; others are system problems such as the environmental risks of growing mega cities, or combinations of processes affecting global and regional climate change.

Monday, November 21, 2011 2:00PM - 2:35PM –
Session J29 Invited Session: Roughness-based Superhydrophobic Surfaces: Fundamentals and Future Directions Ballroom III-IV

2:00PM J29.00001 Roughness-based Superhydrophobic Surfaces: Fundamentals and Future Directions , NEELESH PATANKAR, Department of Mechanical Engineering, Northwestern University — Superhydrophobicity of rough surfaces has attracted global interest through the past decade. There are naturally occurring instances of such surfaces, e.g., lotus leaves, which led to the popular term "lotus effect." Numerous applications in wide ranging areas such as drag reduction, self-cleaning, heat exchangers, energy conversion, condensation, anti-icing, textile, desalination, etc., are being explored by researchers worldwide. The signature configuration for superhydrophobicity has been "bead-like" drops on rough surfaces that roll-off easily. This becomes possible if the liquid does not impale the roughness grooves, and if the contact angle hysteresis is low. Finding appropriate surface roughness is therefore necessary. A thermodynamic framework to enable analysis of this problem will be presented. It will be noted that the success of rough superhydrophobic substrates relies on the presence of gas pockets in the roughness grooves underneath the liquid. These gas pockets could be those of air from the surrounding environment. Current design strategies rely on the availability of air. However, if the rough substrates are fully submerged in the liquid then the trapped air in the roughness grooves may not be sustained. A design approach based on sustaining a vapor phase of the liquid itself in the roughness grooves, instead of relying on the presence of air, will be presented. The resulting surfaces, referred to as vapor stabilizing substrates, are deemed to be robust against wetting transition even if no air is present. Applications of this approach include low drag surfaces, nucleate boiling at dramatically low superheats, among others. The concept can be generalized to other transitions on the phase diagram, thus enabling the design of rough surfaces for phase manipulation in general.

Monday, November 21, 2011 2:40PM - 3:15PM –

Session K27 Invited Session: Experimental Manipulation of Wall-Bounded Turbulence Ballroom I-II

2:40PM K27.00001 Experimental Manipulation of Wall-Bounded Turbulence , BEVERLEY MCKEON, California Institute of Technology — The potential gains in efficiency across a range of applications associated with successful control of wall- bounded turbulence are well-known, as are the many challenges this interdisciplinary problem encompasses. Further, while there have been recent developments in fundamental wall turbulence research, including description of the characteristics of coherent structures (e.g. Adrian, 2007) and the very large scale motions (e.g. Mathis *et al*, 2009), many fundamental questions remain, in particular with respect to scaling behavior as the Reynolds number is increased to practically important values. Progress can be made in both understanding the response of canonical flows to forcing at the wall and isolating mechanisms important to the unperturbed flows using open-loop forcing. To this end, recent work has shown how small amplitude perturbations to surface roughness designed to isolate individual spatial and temporal scales (a "dynamic roughness") can be used to manipulate the characteristics of wall turbulence. This talk will describe the experimental manipulation of a turbulent boundary layer using dynamic roughness and the connection to the spatio-temporal (spectral) information traditionally reported in turbulence studies. The development of tools from earlier work on directional amplification of the Navier-Stokes operator into simple models by which to interpret the coupling between the wall motion and the flow will be described, and some examples of the understanding of structure and Reynolds number trends in the canonical (unperturbed) flow that emerge from the perturbed case will be given. The contribution of students and collaborators, in particular Dr. Ati Sharma (University of Sheffield, U.K.) and Ian Jacobi (Caltech), to this work and the support of AFOSR (grant FA #9550-09-1-0701, program manager John Schmisseur) are gratefully acknowledged.

Monday, November 21, 2011 2:40PM - 3:15PM –

Session K29 Invited Session: DNS and Multi-Scale Modeling of Multi-Phase Flows Ballroom III-IV

2:40PM K29.00001 DNS and Multi-Scale Modeling of Multi-Phase Flows , GRETAR TRYGGVASON, University of Notre Dame — Direct numerical simulations (DNS) of multiphase flows, where every continuum length and time scale is fully resolved, have now advanced to the point where it is possible to study in considerable detail fairly complex systems, such as the flow of hundreds of bubbles, drops, and solid particles. Here we discuss such simulations from a multi-scale perspective, focusing on two aspects: First of all, DNS results can help with the development of closure relations of unresolved processes in simulations of large-scale "industrial" systems. As an example we discuss recent results for deformable bubbles in weakly turbulent channel flows. The lift induced lateral migration of the bubbles controls the flow, but the lift is very different for nearly spherical and more deformable bubbles, resulting in different flow structures and flow rates. Nevertheless, the results show that the collective motion of many bubbles leads to relatively simple flow structure in both cases, emphasizing the need to examine as large a range of scales as possible. The other multi-scale aspect results from the fact that multiphase flows often produce "features" such as thin films, filaments, and drops that are much smaller than the "dominant" flow scales. The geometry of these features is usually simple, since surface tension effects are strong and inertia effects are relatively small. In isolation these features are therefore often well described by analytical or semi-analytical models. Recent efforts to capture thin films using classical thin film theory, and to compute mass transfer in high Schmidt number flows using boundary layer approximations, in combination with direct numerical simulations of the rest of the flow, are described.

3:15PM - 3:15PM –

Session LA Poster Session (3:15-5:00PM) Ballroom I-IV Foyer

LA.00001 Oscillatory Behavior of an Arc Airfoil in Low-Speed Airflow , MAJID MOLKI, NEGIN SATTARI, Southern Illinois University Edwardsville — A computational investigation is conducted to study the oscillatory behavior of an arc airfoil situated in low-speed airflow. The present work is relevant to situations where the conventional rigid airfoils do not apply, such as the flight of bats. The outcome of this study is also beneficial in the design of micro air vehicles with flexible wings. The computations are performed using a deforming mesh to accommodate the airfoil oscillations. An unsteady, spatially second-order algorithm is employed to capture the time-variations of the lift and drag coefficients. A key feature of the present work is the flow response to airfoil oscillations. Fast Fourier Transform was applied to various parameters of the flow. For certain values of angle of attack for the non-oscillating airfoil, the flow has a dominant frequency and a well-defined vortex shedding. For other values of angle of attack, the flow around the non-oscillating airfoil contains many frequencies and has complex vortical structures. However, the oscillating airfoil in all cases makes the flow field periodic with well-defined patterns of vortex shedding. In this work, the flux of vorticity from the airfoil surface into the airflow is computed and compared with the pressure gradient along the surface of the airfoil. Effects of oscillations on magnitude and behavior of aerodynamic forces are also studied.

LA.00002 A universal law of the elasticity of multilamellar lipid membranes under compression , YVES DUBIEF, LEONIE COWLEY, School of Engineering, University of Vermont, Burlington VT — Multilamellar lipid membranes play critical roles in the mechanics and chemistry of cells, lung and biolubrication of articular joints. One of their most interesting mechanical properties is their elasticity than enable them to resist anisotropic compression where the pressure in the direction normal to the membrane is larger than in other directions. This resistance to compression is strongly dependent on hydration, or the number of water molecules confined between two adjacent lipid membranes. Using coarse-grained molecular dynamics, we show that the elastic behavior or multilamellar membranes is in fact universal over a large range of hydration. A universal law is derived from considerations of intermolecular forces and volume vacancy between lipid molecules. The ability of the proposed law to predict the increase of the membrane surface area as a function of the compression rate and hydration is expected to help the modeling of multilamellar membranes at the continuum level.

LA.00003 Designing self-propelling micro-swimmers using responsive gels[1]

, BENJAMIN BINGHAM, HASSAN MASOUD, ALEXANDER ALEXEEV, George W. Woodruff School of Mechanical Engineering, Georgia Institute of Technology — We use computational modeling to design a synthetic micro-swimmer that not only self-propels but also navigates in highly viscous environments. Our simple swimmer consists of a cubic gel body with two rectangular stiff flaps attached to its opposite sides and a stimuli-sensitive flexible flap at the body front. The responsive gel undergoes periodic expansion and contraction that can be induced by certain external stimuli such as temperature, light, magnetic or electric fields. The periodic changes in the volume of the body lead to asymmetric beating motion of the propulsion flaps which propel the micro-swimmer through the inertialess fluid. We study the effect of body elasticity on the locomotion of our swimmer and show how the elasticity of the body can be harnessed to induce forward and backward swimming motion. We also demonstrate that our swimmer can successfully turn in the desired direction following the bending of the responsive steering flap. In this scenario, the steering flap bends and creates flow asymmetry which results in the swimmer rotation.

[1]Financial support from the Georgia Tech President's Undergraduate Research Award (PURA) is gratefully acknowledged.

LA.00004 Flexible Wing Pitch-up and Pitch-down in Free Steam Flows

, D. QI, Western Michigan University, G. HE, Institute of Mechanics Chinese Academy of Sciences, Y. LIU, Sichuan Universirty, China — The direct simulations of the pitch-up and pitch-down of flexible and rigid wings in a free stream are conducted at the Reynolds number of $Re = 100$ by using the lattice Boltzmann flexible particle method. The effect of bending flexibility in span-wise direction on unsteady aerodynamics are investigated. It is found that when the reduced frequency is large, the lift and drag forces increase nonlinearly upto a maximum as the flexibility increases, then falls down as the flexibility becomes excessively large. The maximum value in both lift and drag forces are significantly larger for a flexible wing than for a rigid wing. However, when the reduced frequency is small, no obvious lift maximum is observed. It seems that flexibility can be used to enhance the lift force at a high reduced frequency. The power efficiency, or lift force per input power, has a similar behavior to the lift, indicating the flexibility could benefit the power efficiency. Surprisingly flexibility improves lift only during pitch-down motion while the flexibility has a negative impact on lift during pitch-up motion, indicating that the pitch-down motion dominates the lift improvement due to flexibility. In a maneuver case a small and adequate deformation may largely enhances the wake capture, results in large LEV and TEV and reduces the flow separation. The wake capture is a main mechanism for a flexible wing to improve the lift.

LA.00005 Simplified physical models of the flow around flexible insect wings at low Reynolds numbers

, STEVE HARENBERG, JOHNNY REIS, LAURA MILLER, UNC - Chapel Hill — Some of the smallest insects fly at Reynolds numbers in the range of 5-100. We built a dynamically scaled physical model of a flexible insect wing and measured the resulting wing deformations and flow fields. The wing models were submerged in diluted corn syrup and rotated about the root of the wing for Reynolds numbers ranging from 1-100. Spatially resolved flow fields were obtained using particle image velocimetry (PIV). Deformations of the wing were tracked using DLTdv software to determine the motion and induced curvature of the wing.

LA.00006 Particle Deposition During Airway Closure[1]

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

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

LA.00007 Transient motion of mucus plugs in respiratory airways[1]

, PARSA ZAMANKHAN, YINGYING HU, Biomedical Engineering Department, University of Michigan, BRIAN HELENBROOK, Department of Mechanical and Aeronautical Engineering, Clarkson University, SHUICHI TAKAYAMA, JAMES B. GROTBERG, Biomedical Engineering Department, University of Michigan — Airway closure occurs in lung diseases such as asthma, cystic fibrosis, or emphysema which have an excess of mucus that forms plugs. The reopening process involves displacement of mucus plugs in the airways by the airflow of respiration. Mucus is a non-Newtonian fluid with a yield stress; therefore its behavior can be approximated by a Bingham fluid constitutive equation. In this work the reopening process is approximated by simulation of a transient Bingham fluid plug in a 2D channel. The governing equations are solved by an Arbitrary Lagrangian Eulerian (ALE) finite element method through an in-house code. The constitutive equation for the Bingham fluid is implemented through a regularization method. The effects of the yield stress on the flow features and wall stresses are discussed with applications to potential injuries to the airway epithelial cells which form the wall. The minimum driving pressure for the initiation of the motion is computed and its value is related to the mucus properties and the plug shape.

[1]Supported by HL84370 and HL85156.

LA.00008 Mucus Rupture in A Collapsed airway: An Experimental Study[1]

, YINGYING HU, SHIYAO BIAN, JAMES B. GROTBERG, Department of Biomedical Engineering, University of Michigan — Mucus plugs can completely obstruct an airway. Difficulty in mucus clearance results in lost gas exchange and inflammation. Non-Newtonian properties of mucus, yielding stress and shear-thinning, play significant roles in mucus clearance. We use aqueous carbopol 940 as a mucus stimulant to study clearance of a mucus plug with properties of yielding stress and shear-thinning in a bench-top experiment. A collapsed airway of the 12^{th} generation in a human lung is simulated in a two-dimensional PDMS channel. A stable pressure drop is set along the plug to drive rupture. A micro-PIV technique is used to acquire velocity fields during the rupture process. A yielding pressure drop (initiating plug yielding) is nearly independent of initial plug length. Plug rupture can occur by focused deformation along the centerline or by total plug propagation where the trailing film is thicker than the precursor film. Maximum velocity appears at the rupture moment, and increases at higher pressure drop or smaller plug length. The wall shear gradient can undergo a rapid reversal when rupture occurs, possibly an injurious event to underlying airway epithelial cells.

[1]This work is supported by NIH: HL84370 and HL85156.

LA.00009 Microbubble array under ultrasound excitation and its application in cell capture

, JIFU TAN, ANTONY THOMAS, YALING LIU, Lehigh University — In this paper, we studied the fluid flow and particle motion induced by a microbubble array exited under ultrasound. A circulation vortex generated in spaces between microbubble array could be controlled by the spacing of the array, the amplitude and frequency of the ultrasound. We observed that particles/cells of particular size could be trapped near the microbubble surface. The microbubble array is used to enrich bimolecular concentration locally as well as capture cells. Such active attraction mechanism has advantages over traditional passive capture methods.

LA.00010 Dynamic behavior of lean swirling premixed flame generated by change in gravitational orientation -Nonlinear forecasting based on the complex network theory, MASAHITO AMANO, KOSHIRO MAKI, TAKAYA MIYANO, HIROSHI GOTODA, Ritsumeikan University — We experimentally investigated the deterministic nature in the dynamic behavior of a lean swirling premixed flame generated by a change in gravitational orientation from the viewpoint of nonlinear forecasting based on the complex network theory. When the gravitational direction is changed relative to the flame front, i.e., in inverted gravity, an unstable flame is formed in a limited domain of equivalence ratio and swirl number (Gotoda. H et al., PRE, vol. 81, 026211, 2010). A radial basis function network, which quantifies the predictability of time variations in flame front fluctuations of the unstable flames, is applied as a sophisticated nonlinear forecasting method in this work. A nonlinear forecasting approach based on the complex network theory clearly demonstrates that the dynamic behavior of flame front fluctuations represents low-dimensional deterministic chaos. The deterministic nature in the dynamic behavior of flame front fluctuations revealed using the radial basis function network has not yet been reported in previous research on the flame front instability.

LA.00011 Analysis and Development of a Quick Acting Diaphragm-less Shock Tube Driver[1], ROCCO PORTARO, HOI DICK NG, Concordia University, Montreal, Canada — This work discusses the construction and performance characteristics of a diaphragmless shock tube driver. Shock waves play integral roles in many industrial, medical and scientific environments, consequently it is important to observe the behaviour of these waves and how they interact with their surroundings. The diaphragmless shock tube provides a quick and effective means of producing shock waves in gases. The major advantages compared to conventional diaphragms include, minimal downtime between repeated experiments, opening times comparable to those of conventional diaphragms and infinitely adjustable opening pressure without the use of various diaphragm thicknesses. Moreover, the diaphragmless design also eliminates fragments that are carried downstream of the shock tube once the conventional diaphragm is ruptured. The design utilized in this work is built on that of Downey et al. [M.S. Downey, T.J. Cloete, A.D.B.Yates, Shock Waves 21(4): 315-319, 2011] and is improved in order to obtain faster opening times leading to stronger shock formation. Furthermore in depth numerical analysis using the commercial CFD package Fluent is carried out to validate experimental data for driven pressures and opening times as a function of driver pressure.

[1]Supported by Fonds de recherche du Quebec - Nature et technologies

LA.00012 ABSTRACT WITHDRAWN —

LA.00013 Calibration of the k-ϵ model constants for use in CFD applications, NINA GLOVER, SERGE GUILLIAS, LIORA MALKI-EPSHTEIN, University College London — The k-ϵ turbulence model is a popular choice in CFD modelling due to its robust nature and the fact that it has been well validated. However it has been noted in previous research that the k-ϵ model has problems predicting flow separation as well as unconfined and transient flows. The model contains five empirical model constants whose values were found through data fitting for a wide range of flows (Launder 1972) but ad-hoc adjustments are often made to these values depending on the situation being modeled. Here we use the example of flow within a regular street canyon to perform a Bayesian calibration of the model constants against wind tunnel data. This allows us to assess the sensitivity of the CFD model to changes in these constants, find the most suitable values for the constants as well as quantifying the uncertainty related to the constants and the CFD model as a whole.

LA.00014 ABSTRACT WITHDRAWN —

LA.00015 Control of Polymer Electrodes Shape via Microflow Control in a Drying Droplet[1], YUNSEOK JANG, Korea Institute of Machinery & Materials — We demonstrated a simple patterning method for the deposition of polymer electrodes such as poly(3,4- ethylenedioxythiophene)/poly(styrenesulfonate) (PEDOT/PSS). We made use of the difference in wettability between hydrophobic surfaces and hydrophilic surfaces to make the patterns. However, the patterns made with our patterning method created undesirable ring-like stains, which were caused by the outward flow of the solute within the PEDOT/PSS solution drop. To achieve homogenous device performance, we proposed a simple process for removing this ring-like stain by making the surface tension gradient in the PEDOT/PSS solution drop. Because this surface tension gradient causes the inward flow of the solute within the PEDOT/PSS solution drop, the ring-like stain is removed. Finally, we confirmed the potential of our patterning method for polymer electrodes such as the PEDOT/PSS by fabricating pentacene thin-film transistors (TFTs) and measuring the electrical properties of the pentacene TFTs.

[1]This study was supported by a grant (B551179-08-03-00/ B551179-10-01-00/ NK162D/ NK162H) from the cooperative R&D Program funded by the Korea Research Council Industrial Science and Technology, Republic of Korea.

LA.00016 Experimental investigation of free falling thin disks. Part II: Onset of three dimensional motion from zigzag to spiral, CUNBIAO LEE, College of Engineering, Peking University, HONGJIE ZHONG — The free falling motion of thin disk with small dimensionless moment of inertia was investigated experimentally. Transition from two dimensional zigzag motions to three dimensional spiral motions takes place as a consequence of breakup of planar symmetry flow structures. The counter-rotating vortices behind the disk became asymmetry induced by the instability of three dimensional disturbances, one of the separation location moves from the edge to the disk broad surface which induced a new upright vortex. The oscillation in the direction normal to zigzag plane increases with the development of instability. Meanwhile, the oscillation of nutation angle vanishes and reaches a constant value. In addition, we have shown that small moment of inertia is responsible for growth of disturbances in the third dimension by means of generalized Kirchhoff equations.

LA.00017 Experimental study on thermocapillary motion of isolated drop and coalescence problems of drops, JINGCHANG XIE, Institute of Mechanics, Chinese Academy of Sciences, HAI LIN, Institute of Mechanic, Chinese Academy of Sciences — Thermocapillay migrations of drops under temperature gradient were studied through ground-based experiment, experiment using drop tower and space experiment in microgravity. The motion of isolated drop at moderate to large Marangoni numbers (Ma) and the interaction of drops were investigated. Experimental data show that the scaled migration velocity of isolated drop, V/V_{YGB}, appears an obvious decrease trend with the increase of Maragoni number up to 5500. This result does not agree with some theoretical predictions. Interferometry was applied in our space experiment to visualize the whole temperature field and to get detailed informations of temperature variation around a moving drop and the thermal wake behind it. Interferometric images indicate that drop's migration very sensitively follows the direction of temperature gradient because of slow migration velocity and microgravity condition. The temperature disturbance around a leading drop and the thermal wake behind it would exist for a quite long time in the real case. The variation of temperature field would substantially affect the migration velocity of a trailing drop in both direction and value, and this would bring about coalescence problems of two or multiple drops.

LA.00018 Droplet Breakup Mechanisms in Air-blast Atomizers , AMIR ABBAS ALIABADI, PhD Candidate Mechanical Engineering University of British Columbia, SEYED MOHAMMAD TAGHAVI, PhD Candidate Chemical Engineering University of British Columbia, KELLY LIM, BASc Candidate Mechanical Engineering University of British Columbia — Atomization processes are encountered in many natural and man-made phenomena. Examples are pollen release by plants, human cough or sneeze, engine fuel injectors, spray paint and many more. The physics governing the atomization of liquids is important in understanding and utilizing atomization processes in both natural and industrial processes. We have observed the governing physics of droplet breakup in an air-blast water atomizer using a high magnification, high speed, and high resolution LASER imaging technique. The droplet breakup mechanisms are investigated in three major categories. First, the liquid drops are flattened to form an oblate ellipsoid (lenticular deformation). Subsequent deformation depends on the magnitude of the internal forces relative to external forces. The ellipsoid is converted into a torus that becomes stretched and disintegrates into smaller drops. Second, the drops become elongated to form a long cylindrical thread or ligament that break up into smaller drops (Cigar-shaped deformation). Third, local deformation on the drop surface creates bulges and protuberances that eventually detach themselves from the parent drop to form smaller drops.

LA.00019 Development of Anodized-Aluminum Temperature-Sensitive Paint for Flow Field Measurements , AKIHISA AIKAWA, Kyushu University, HIROTAKA SAKAUE, JAXA — Anodized-aluminum temperature-sensitive paint (AA-TSP) is developed to capture the flow fields related to the temperature. Instead of using conventional polymer type TSP, the anodized-aluminum coating can hold material properties from cryogenic to high temperatures limited by its melting point of $\sim$ 930 K. We studied various luminophores onto this coating as a global temperature sensor. Six different quantum dots, which varied the luminescent peak wavelengths, are applied onto this coating. Rhodamine-B, pyronin B, pyronin Y, and europium complex are studied, which are also good candidates as the temperature sensing probes. The temperature calibrations of the developed AA-TSP are shown. The temperature is varied from 100 to 500 K. The temperature sensitivities of the developed AA-TSPs are related to the temperature range calibrated. Comparisons of the AA-TSPs related to the sensitive temperature range are included. A global temperature measurement in a hypersonic wind tunnel is shown as a demonstration of the developed AA-TSPs.

LA.00020 Dual luminescence imaging applied for capturing the temperature distribution of a super-cooled droplet in collision icing , KATSUAKI MORITA, KOJI OKAMOTO, The University of Tokyo, HIROTAKA SAKAUE, Japan Aerospace Exploration Agency — Dual luminescence image is applied to capture the temperature distribution of a super-cooled droplet in icing when the droplet collides onto a plate. The imaging technique captures the temperature-sensitive luminescence and the temperature-insensitive luminescence, which are spectrally separated. These images are captured by a hi-speed color camera. The icing process from super-cooled condition to ice gives insight into further understandings of the icing in flights, power cables, architectures, etc. The icing formation is shown by time steps by using the hi-speed camera. The formation is discussed from the images captured. The plates are coated with icephobic coatings to understand their effects on the collision icing.

LA.00021 ABSTRACT WITHDRAWN —

LA.00022 Influence of local parameters on the dispersion of traffic-related pollutants within street canyons , STYLIANI KARRA, LIORA MALKI-EPSHTEIN, UCL, MARTIN HYDE COLLABORATION[1] — Ventilation within urban cities and street canyons and the associated air quality is a problem of increasing interest in the last decades. It is important for to minimise exposure of the population to traffic-related pollutants at street level. The residence time of pollutants within the street canyons depends on the meteorological conditions such as wind speed and direction, geometry layout and local parameters (position of traffic lane within the street). An experimental study was carried out to investigate the influence of traffic lane position on the dispersion of traffic-related pollutants within different street canyons geometries: symmetrical (equal building heights on both sides of the street), non-symmetrical (uniform building heights but lower on one side of the street) and heterogeneous (non-uniform building heights on both sides of the street) under constant meteorological conditions. Laboratory experiments were carried out within a water channel and simultaneous measurements of velocity field and concentration scalar levels within and above the street canyons using PIV and PLIF techniques. Traffic -related emissions were simulated using a line emission source. Two positions were examined for all street geometries: line emission source was placed in the centre of the street canyon; line emission source was placed off the centre of the street.

[1]TSI Incorporated

LA.00023 Wake Structure of Oscillating Cylinders , MICHAEL COHRS, WAYNE ERNST, ASHWIN VAIDYA, Montclair State University — We studied the vortex-induced oscillations of a tethered cylinder in a flow tank. The cylinder shows various changes (steady, periodic, autorotation) in its orientation as a function of Reynolds number and particle aspect ratio. In particular, we examine a new metric- namely, the distance from the cylinder to the vortex as a function of the Reynolds number. This simple metric proves to be very helpful in characterizing the changes in vortex structure and serves as a useful marker for the various bifurcations in the flow. It is implied through data visualization that this critical point (which indicates a change in wake structure) is present regardless of the lengths of the cylinders. We examined Reynolds numbers in the range of about Re = 550 to Re = 4800, based on the cylinder's length. We also examined relationships between Strouhal number for the particle and associated vortex as a function of Reynolds number and particle aspect ratio.

LA.00024 The mass, energy, space and time systemic theory- Black hole will take the dark comet to impact our earth , DAYONG CAO, Beijing Natural Providence Science & Technology Development Co., Ltd — Our solar system has a companion black hole-"Tyche." A black hole have many dark planets (dark comets) like that Sun have many planets. The companion black hole could take its dark comets to go into our solar system, So there are a lot of dark comet near the orbit of Jupiter. The companion black hole go near our solar system every 25-27 million years. It could take its dark comets to impact our earth and could trigger the Mass Extinction. The black hole is the space-time center like a dark gas ball. So it absorb the space-time to go to its center like that Sun absorb the mass of matter by the gravity. Because it can absorb the space-time, when the companion black hole go near our solar system, it can cause the expand of space-time of solar system and the red-shift of our observation. The new idea will instead of the "cosmic inflation". The black hole and sun build up a system together. When the black hole inbreak our solar system in 20 years, not only it will break our environment, but aslo it will change the systemic genetical code of our life and thing. So it will kill many lives and will produce new life. So it could trigger the Mass Extinction.

LA.00025 The mass, energy, space and time systemic theory-MEST , DAYONG CAO, Beijing Natural Providence Science & Technology Development Co., Ltd — The displacement and period of the orbit of the motion are the space-time; the probability of them are the quantum space-time. Both of the energy-momentum tensor and metric tensor belong to the gravitational field. It can direct the space-time. The black hole has a space-time center as a origin of a mass-energy coordinate system and a binding energy of space-time like that sun has a mass-energy center and the nuclear energy. And it has a dark matter field around. There is a balance energy equation between sun and black hole.

$$E = h\nu = mc^2 \tag{1}$$

Among it, E: the energy of wave of sun, m: the mass of wave, c: the velocity of wave, ν: the frequence of wave, h: the Planck constant.

$$E'\psi = i\hbar \frac{\partial \psi}{\partial t} \tag{2}$$

$$m'\psi = -i\hbar \frac{\partial \psi \partial t}{(\partial x)^2} \tag{3}$$

Among it, $E'\psi$: the energy of dark wave of black hole, $m'\psi$: the mass of dark wave, c$'$: the velocity of dark wave, ψ: the Wave Functions.

$$h\nu + E'\psi = mc^2 + m'\psi c'^2, (c'^2 = -\frac{(\partial x)^2}{(\partial t)^2}) \tag{4}$$

LA.00026 ABSTRACT WITHDRAWN —

LA.00027 Axially asymmetric rotating tank experiments for thermally forced stationary waves in geophysical fluids , HUEI-PING HUANG, Arizona State University, JULIAN HUNT, University College London, ASHISH SHARMA, LOUIS TSE, Arizona State University, HARINDRA FERNANDO, University of Notre Dame, ANDREY GUNAWAN, PATRICK PHELAN, ARTHUR MADRID, MICHAEL THOMPSON, Arizona State University — Fluid dynamical experiments using a rotating tank with an imposed radial temperature gradient provide classical examples of the realization of large-scale atmospheric circulation in a laboratory setting. The last decade has seen a revival of such experiments for research and education. Classical rotating tank experiments have adopted axially symmetric boundary conditions to maintain a zonally uniform "pole-to-equator" temperature gradient. Symmetry breaking arises from internal dynamics of baroclinic instability. A notable exception is the class of experiments for topographic effects, in which an isolated obstacle is added to the bottom boundary. This study explores a new type of experiments that are axially asymmetric due to an imposed, zonally non-uniform, temperature in the lateral boundary. This mimics the contrast of warm pool versus cold tongue in the tropical Pacific Ocean. The experiments produced thermally forced quasi-stationary waves with their wavelength comparable to the scale of the localized thermal boundary forcing. The applications of this new type of experiments to understanding Earth's climate will be discussed.

LA.00028 Possible Electrostatic Precursors to Granular Slip Events , TROY SHINBROT, NIRMAL THYAGU, Rutgers University — For at least the past 40 years, reports have repeatedly appeared of atmospheric lightning and related effects preceding major earthquakes. Many of these reports are anecdotal and of uncertain reliability, while others appear to have been substantiated by more recent scientific electric field measurements. In this work, we describe laboratory experiments that appear to exhibit sporadic, but statistically significant, electrical precursors to granular slip events. The cause of this phenomenon is unclear: the materials used are neither piezoelectric nor triboluminescent. We speculate that the electrical signals may be related to other electrical phenomena known to be associated with material failures in other contexts, for example in crack formation and in tape peeling.

LA.00029 The formation of bulges and coastal currents from buoyant outflows , PABLO HUQ, University of Delaware — Buoyant outflows exiting (e.g. from an estuary or a sea strait) to the coastal ocean may form large bulges and coastal currents which propagate downshelf. Laboratory experiments show that the effect of the earth's rotation is to deflect the trajectory of the buoyant outflow into an inertial circle so that the buoyant outflow impacts the coastline further downshelf at an impact angle ϕ. Bulges form if ϕ is larger than 60°. Bulges store a significant fraction of the buoyant outflow; this attenuates the scales of the downshelf coastal current. The characteristics of the coastal current also depend on the ambient depth parameter, h/H, and the bottom slope parameter R/y$_B$. The width of the outflow also influences whether or not bulges form; bulge formation is described by a two-parameter space comprising the ambient depth parameter h/H and the Kelvin number K. Comparisons are made with oceanic observations.

LA.00030 Lattice Boltzmann inverse modeling of water flows in the Everglades National Park , ANDREW PEARSON, MICHAEL SUKOP, Florida International University, VICTOR ENGEL, South Florida Natural Resources Center, Everglades National Park — Knowledge of water flow in the Everglades National Park is vital for ecosystem restoration efforts. Of interest here is the planned rehabilitation of "ridge-and-slough" habitats, characterized by numerous adjacent sawgrass ridges aligned parallel to the flow direction. In many areas, reduced water flow due to water management systems has resulted in the conversion of ridge-and-slough landscapes into dense sawgrass stands. Previous studies of water flow have included tracer experiments, typically performed at several points in the region, with measurements taken over several days. Here we report initial results of inverse modeling of data from the EverTREx series of experiments using the parameter estimation code PEST coupled with a Lattice Boltzmann code. We perform two-component simulations in two dimensions, using satellite imagery to model the presence of vegetation in analogy with a porous medium. The final goals of this project are to reduce or remove the need for on-the-ground experiments for determining water flow characteristics in the Everglades, and to predict the effect of changes in water management systems on the flow.

LA.00031 Effective Interactions Between Intruders in Vibrated Granular Materials , RACHEL DERBY, University of Maryland, Baltimore County, BRIAN UTTER, James Madison University — Despite the strong fluctuations in a rapidly moving granular material, dissipation and correlations in collisions can lead to long range forces in granular materials. In this experiment, we study the long-range attraction between two objects when immersed in a vibrated granular system. Depending on the strength of vibration, a granular system can take the form of a gas or be fluidized. We place two large intruders in each of these systems to track the effective interactions between the intruders, varying frequency, amplitude, size of grains, and the shape of the intruder. In particular, we study the interaction of spheres in a granular gas and the effective force between plates in a granular fluid. Using image processing, we track the separation of the intruders over time. We find that parallel plates attract if they are separated by less than approximately 15 particle diameters and reach a final position in which one ordered layer of grains is between them. Granular gas experiments suggest a long-range interaction, but observed effects in the mean position are much weaker than the fluctuations. Ongoing work focuses on varying the vibration parameters (amplitude, frequency, & waveform) and increasing statistics.

LA.00032 Effects of Shear Thickening Properties in Oscillating Non-Newtonian Fluids[1] , ROXANNE ABLE, YOGI PATEL, JON BOUGIE, Loyola University Chicago — We study the behavior of a shear-thickening, non-Newtonian fluid when shaken at a variety of accelerational amplitudes and frequencies. Mixing corn starch with water, using cesium chloride to match density and prevent separation, produces a fluid with shear thickening properties. When a thin layer of this fluid is vertically oscillated, it can produce Faraday waves as well as other phenomena that are characteristic of non-Newtonian fluids, such as stable holes and time-dependent, delocalized regions that grow from small initial disturbances in the fluid layer. We investigate how the concentration of corn starch (and as a result the shear-thickening properties of the fluid) affects which phenomena are observed, and we demonstrate that this concentration does have a significant effect on the fluid behavior.

[1]This research was supported by Research Corporation for Science Advancement.

LA.00033 Pulsed laser induced self-assembly of nanoparticle arrays: Competing liquid phase instabilities , J.A. DIEZ, A.G. GONZALEZ, IFAS, Universidad Nac. del Centro de la Prov. de Buenos Aires, Tandil, Argentina, Y. WU, Department of Materials Science and Engineering, University of Tennessee, Knoxville, TN, J.D. FOWLKES, Center for Nanophase Materials Sciences, Oak Ridge National Laboratory, Oak Ridge, TN, N.A. ROBERTS, P.D. RACK, Department of Materials Science and Engineering, University of Tennessee, Knoxville, TN, L. KONDIC, Dept. of Mathematical Sciences, Center for Applied Mathematics and Statistics, New Jersey Inst. of Technology, Newark, NJ — Thin film copper rings were synthesized on silicon dioxide thin films with various radii, thicknesses and widths and were subsequently liquefied via a nanosecond pulse laser treatment. During the nanoscale liquid lifetimes, the rings experience competing retraction dynamics and thin film and/or Rayleigh-Plateau type of instability, which lead to arrays of ordered nanodroplets. Ultimately, the original geometry dictates the instability pathway, which for narrow rings obeys the Rayleigh-Plateau type of instability, while for wider rings is influenced by the thin film instability. Hydrodynamic simulations describe well the observed time and length scales as well as the observed radial and circumferential instabilities which lead to different nanodroplet ensembles.

LA.00034 Effect of viscosity ratio on liquid-liquid jets subjected to radial/axial electric field , SIDDHARTH GADKARI, IITB-Monash Research Academy, IIT Bombay, Powai, India, ROCHISH THAOKAR, Department of Chemical Engineering, IIT Bombay, Powai, India — Linear stability analysis for viscous liquid-liquid electrified jets subjected to axisymmetric (m=0) and asymmetric (m=1) perturbations has been performed. Both radial and axial electric field configurations are considered and the importance of viscosity ratio (λ= viscosity of surrounding fluid/Viscosity of inside fluid) is studied. λ is shown to have a damping effect on both the modes of perturbation. However the effect is more pronounced for the m=1 mode as compared to m=0 mode in the presence of electric fields. Viscosity ratio, along with electric field can control the dominance of individual modes. While m = 1 mode can only be realized in the lower λ limit when radial field is on, it is always possible to realize m = 1 mode under axial fields provided the threshold field is applied. A phase diagram showing predominance of the two modes at any given value of electric field and viscosity ratio is generated for both axial and radial electric field setups. This phase diagram can serve as a guideline for the required set of operating parameters in order to suppress/realize any desired mode of instability.

LA.00035 Kinetic approach to Kaluza's magnetohydrodynamics , A. SANDOVAL-VILLALBAZO, Department of Physics and Mathematics, Univeridad Iberoamericana, Ciudad de México, L.S. GARCIA-COLIN, Department of Physics, UAM-I — Ten years ago we presented a formalism by means of which the basic tenets of relativistic magnetohydrodynamics were derived using Kaluza's ideas about unifying fields in terms of the corresponding space time curvature for a given metric.[1] In this work we present an attempt to obtain the thermodynamic properties of a charged fluid using using Boltzmann's equation for a dilute system adapted to kaluza's formalism. The main results that we obtain are analytical expressions for the main currents and corresponding forces, within the formalism of linear irreversible thermodynamics. We also indicate how transport coefficients can be calculated. Other relevant results are also mentioned.

[1]A. Sandoval-Villalbazo and L.S. Garcia-Colin; Phys. of Plasmas 7, 4823 (2000).

LA.00036 Effect of electrode geometry on charging of a water droplet in a dielectric liquid[1] , MYUNGMO AHN, DO JIN IM, IN SEOK KANG, POSTECH — Electrophoresis of a charged droplet (ECD) in a dielectric liquid can be used as a droplet manipulation method without complicated electrode or circuit design. For further utilization of its advantages, we need thorough understanding on the charging mechanism and the electrode geometry effect. We have investigated the effect of electrode geometry on the amount of charging experimentally and numerically. In the experiments, we have checked the difference of charge amount between the pin type and the plate type electrodes. A high speed camera and an electrometer are used for the measurement simultaneously. We also have calculated the charge amounts for both cases numerically. The charge amounts predicted from the simulation show good agreement with those of the experiments. In both cases, a droplet gets more charges on the pin type electrode. This result can be used for better design in the microfluidic applications using electric charging method.

[1]This work has been supported by Grant R01-2010-0027653 from the National Research Foundation (NRF) of Korea and by the BK21 program of the Ministry of Education, Science and Technology (MEST) of Korea.

LA.00037 A microfluidic platform for impedance analysis and characterization of human umbilical vein endothelial cells , VANESSA VELASCO, KENNY KING, ROBERT KEYNTON, STUART WILLIAMS, University of Louisville — The characterization of endothelium morphology and permeability under fluid shear stress can provide essential information regarding the onset of different pathological conditions, as well as, the uptake of drugs and biomolecules. Real-time assessment of the integrity of cell monolayers and cell motility has been accomplished by implementing electrical impedance analysis. In this study, we report a micro-electrode array biosensor incorporated into a microfluidic platform for the impedance spectroscopy of Human Umbilical Vein Endothelial Cells (HUVECs) under physiological fluid shear stress. The biosensor consists of two adjacent and identical microfluidic channels which allows for simultaneous assessment of the electrical properties of a HUVEC monolayer and that of the cell culture media alone. Additionally, the biosensor is attached to a custom designed electronic system that simplifies the data acquisition and analysis of time-dependent multiplexed measurements for the different electrodes. The impedance spectrum (40 Hz to 10 MHz) was collected for HUVECs before and under different physiological fluid shear stress forces to yield insight to flow induced changes in HUVEC morphology, permeability, and relevant electrical parameters. This biosensor can be used as an *in vitro* model of the endothelium for new drug discovery, as well as, part of biochemical assays for cell mechanism studies.

LA.00038 Non-Newtonian behavior of magnetized ferrofluids as revealed by high speed x-ray phase contrast imaging[1] , JOY A. PRESCOD, A. CALI, S. NUNEZ, R. SMITH, M. VIEIRA, A.D. TRUBATCH, P. YECKO, Montclair State University, Montclair NJ 07043, W.-K. LEE, Advanced Photon Source, Argonne National Laboratory — Objects moving through a magnetized ferrofluid experience enhanced drag as a result of the presence of magnetic particles and magnetic particle agglomerations which form due to magnetic attractive forces. The precise impact of an agglomeration on an object depends on the characteristics of the agglomeration, the relative sizes of the object and agglomeration, as well as other control parameters. In this study, high speed phase contrast imaging was used to directly image the impact of long thread-like magnetic particle agglomerations on the rheological properties of ferrofluids. Particularly, numerous types of interactions between these threads and translating objects, including free-falling 500 micron sized solid glass spheres and intermittently rising vapor bubbles were quantified. At these scales, objects may bind to particle threads resulting in momentary re-direction or arrest of the object's trajectory, alluding to a form of yield stress. Therefore, there is a macro-viscosity property in flows of this type, which has a potentially significant impact of the use of ferrofluids in micro-fluidics and drug delivery.

[1]Supported by NSF grant no. MPS-1016383.

LA.00039 DNA Size Segragation at a Convergent Stagnation Point , JAMES BOULGER, JENNIFER KREFT PEARCE, The University of Texas at Tyler — We investigate using counter-rotating vortices and a convergent stagnation point to segregate different sizes of DNA using a lattice Boltzmann based simulation with a bead-spring model for the polymer. We find that longer DNA molecules are left rotating in the fluid while shorter molecules aggregate at the stagnation point. The separation of the polymers depends on the fluid flow rate and a non-specific attractive force with the channel walls. We report on the robustness of this technique.

LA.00040 Falling ball viscometry of magnetized ferrofluids[1] , ALEX CALI, Montclair State University, W.K. LEE, Advanced Photon Source, Argonne National Laboratory, SAMUEL NUNEZ JR., JOY PRESCOD, ROSE SMITH, A.D. TRUBATCH, MATTHEW VIEIRA, PHILIP YECKO, Montclair State University — Falling spheres of 500 μm were used to perform viscometric experiments on magnetized ferrofluids. The role of the angle of orientation of an applied unifom field relative to the direction of fall has been examined with high speed phase contrast imaging using the Advanced Photon Source. The magnetized ferrofluid exhibits an anisotropic viscosity that we can quantify in terms of a tensorial viscosity coefficient. We find that the effective drag is greater when the fall occurs normal to the applied field rather than parallel to it, a result that is opposite to what is predicted by many ferrofluid magnetoviscosity models, but consistent with the properties of electro- and magneto-rheological fluid, liquid crystal, and polymer fluid rheology models. Finally, we discuss the dispersion of these results in terms of thread-like aggregations of magnetic particles observed in the experiments.

[1]supported by NSF grant no. MPS-1016383

LA.00041 Ink transfer mechanism for gravure-offset printing by using flow visualization , SEUNG-HYUN LEE[1], KI-SANG NAM, Korea Institute of Machinery and Materials — We investigated ink transfer mechanism for gravure-offset printing by monitoring ink transfer process. For convenient visualization of ink transfer, cantilever-type roll-to-plate gravure offset printing system is designed. We visualized ink transfer process from patterned plate to rolling blanket roller. Serial images were captured by using high-speed CMOS camera and long range microscope, and analyzed by image process. We investigated the rotational effect of blanket roller on ink transfer mechanism by comparing the ink transfer process with different pattern angle and printing speed.

[1]Corresponding author

LA.00042 Enhanced evaporation from surfaces of width comparable to that of the mass boundary layer thickness , KARINE IP, SHREYAS MANDRE, Brown University — We investigate the optimal distribution of spacing and size of stomata on a leaf, which enhances net evaporation rate. Although evaporation flux is known to be maximal at contact lines, is diffusion of the vapor also enhanced by the accompanying increase in the total perimeter when the contact lines are adjacent? The evaporation rate from the stomata is analyzed using a 1-D model for stomata distributions on a leaf and a 2-D convection-diffusion equation in the surrounding space. A universal behavior is observed as a function of the Peclet number Pe and stomata size. For stomata much wider than the mass boundary layer thickness, evaporation rate increases as the stomata are made smaller; but below a critical size, the evaporation rate saturates to a constant value. This transition occurs when stomata size is comparable to the mass boundary layer thickness. We experimentally tested this behavior with liquid evaporating from micro-channels of varying channel widths in a wind tunnel providing a range of Pe.

LA.00043 Effects of Microstructural Parameters on Permeability of Fibrous Materials to Non-Newtonian Fluids , B. EMAMI, H. VAHEDI TAFRESHI, Virginia Commonwealth University — In this work, a series of numerical simulations has been devised to relate the non-Newtonian permeability constant of a fibrous medium to its Newtonian counterpart. Developing virtual 3-D geometries that resemble the internal microstructure of a fibrous material, we studied the effects of fiber diameter, fiber in-plane and through-plane orientations, and media's porosity on the media's permeability to Non-Newtonian fluids such as blood. The results of our digital experiments are used in conjunction with available analytical relations, derived to predict the non-Newtonian permeability constants of granular beds, to develop new empirical correlations for fibrous materials.

LA.00044 Time resolved measurements of particle lift off from the wall in a turbulent water channel flow , RENE VAN HOUT, Technion-Israel Institute of Technology, BORIS RABENCOV, JAVIER ARCA, Technion - Institute of Technology — Time-Resolved Particle Image Velocimetry (TR-PIV) and digital holography measurements were carried out in a dilute particle-laden flow tracking both Polystyrene Spheres (PS, ~0.583 mm, d^+ ~10) as well as resolving the instantaneous velocity field of the turbulent flow. Measurements were performed in a closed loop, transparent, square channel facility (50x50 mm^2) at 127.5cm from the inlet with bulk water velocity 0.3 m/s ($\mathrm{Re}_h = 7353$) and friction velocity 0.0174 m/s. Data were captured at 1 kHz, corresponding to a time scale 5x smaller than the flow's viscous scale. Single view digital holographic cinematography was used to track the 3D PS motion inside the VOI (17x17x50 mm^3) including the wall bottom. TR-PIV in a vertical plane (29.3x29.3 mm^2) oriented along the channel's centerline imaged PS together with flow tracers. Discrimination was based on their size difference. Instantaneous sequences of PS plotted on the spatial velocity, vorticity and swirling strength maps showed the effect of turbulent flow structures and resulting particle movement. Results are presented for particles that lift off from the bottom wall as a result of complex interaction with ejection and sweep motions.

LA.00045 Lagrangian particle tracking in strained and buoyant flows , ARMANN GYLFASON, DIMITRY IVANOV, LAHCEN BOUHLALI, Reykjavik University, CHUNG-MIN LEE, California State University Long Beach — We present experimental particle tracking results in two turbulent flow configurations, straining flow and buoyant flow. Our focus is on the influence of large scale structure of such flows on the motions of passive and inertial particles, in particular acceleration statistics, Lagrangian structure functions, and two point statistics. The results are contrasted with previous work, and our results from approximately isotropic flows. The Eulerian structure of the underlying flow field is measured by applying the PIV method.

LA.00046 Pore-scale Analysis of the effects of Contact Angle Hysteresis on Blob Mobilization in a Pore Doublet , SHAO-YIU HSU, ROLAND GLANTZ, MARKUS HILPERT, Johns Hopkins University — The mobilization of residual oil blobs in porous media is of major interest to the petroleum industry. We studied the Jamin effect, which hampers the blob mobilization, experimentally in a pore doublet model and explain the Jamin effect through contact angle hysteresis. A liquid blob was trapped in one of the tubes of the pore doublet model and then subjected to different pressure gradients. We measured the contact angles (in 2D and 3D) as well as the mean curvatures of the blob. Due to gravity effects and hysteresis, the contact angles of the blob were initially (zero pressure gradient) non-uniform and exhibited a pronounced altitude dependence. As the pressure gradient was increased, the contact angles became more uniform and the altitude dependence of the contact angle decreased. At the same time, the mean curvature of the drainage interface increased, and the mean curvature of the imbibition interface decreased. The pressure drops across the pore model, which we inferred with our theory from the measured contact angles and mean curvatures, were in line with the directly measured pressure data. We not only show that a trapped blob can sustain a finite pressure gradient but also develop methods to measure the contact angles and mean curvatures in 3D.

LA.00047 ABSTRACT WITHDRAWN —

LA.00048 Effect on Turbulence Radiation Interaction in Particle Laden Flow[1] , MATHEW CLEVELEAND, SOURABH APTE, TODD PALMER, Oregon State University — The effects of Turbulence Radiation Interactions (TRI) in particulate laden flows can significantly influence thermal radiation fields and corresponding material heating. Most combustion problems contain strong heterogeneities which can be treated stochastically. In pulverized coal combustion these heterogeneities include particulate such as coal, fly-ash, and char. This work expands upon a simplified test case, developed by Deshmukh et al. (Proc. Comb. Inst., 2009), to highlight the effects of fuel particulate on TRI phenomena. This includes the decaying isotropic turbulence with a single-step reaction in the presence of solid particulates. Three different test cases were studied: a simplified gas-phase combustion problem without particulate, non-reacting particulate in the gas-phase combustion, a solid-fuel/gas-oxidizer combustion model. Sensitivity of TRI uncertainties such as thermal radiation fields and thermal emission to the presence fuel particulates is investigated in this test problem using direct numerical simulation for the gas-phase and Monte-Carlo approach for the radiative heat transfer.

[1]This work was funded by DOE-NETL, program manager Cathy Summers.

LA.00049 Shooting inertial wave packets into rotating turbulence , ERAN SHARON, EHUD YAROM, KOBI COHEN, The Hebrew University of Jerusalem — We experimentally study the effect of localized fluctuations in energy injection rate on existing rotating turbulent flow. We show that such fluctuations generate packets of inertial waves, propagating along the axis of rotation. Close to the energy source the propagating pulses leave the steady turbulent field unaffected. Energy is transferred from the pulses into the steady turbulent field only at height h above the injection plane. This height is determined by equating the propagation time of an inertial wave to h, with the time for nonlinear interaction between the wave and the turbulent field. This mechanism, which allows "shooting" turbulence to selected heights, is expected to be important in rapidly rotating geophysical flows.

LA.00050 Flow Instability in Channel Flow with a Streamwise-periodic Array of Circular Cylinders[1] , KYONGJUN LEE, Inha University, Korea, DONG-HYEOG YOON, Korea Institute of Nuclear Safety, KYUNG-SOO YANG, Inha University, Korea — A parametric study has been carried out to elucidate the characteristics of flow instability in laminar channel flow with a streamwise-periodic array of circular cylinders. This flow configuration is relevant to heat exchanger applications. The presence of cylinders in channel flow causes the attached wall boundary layer to separate, leading to a significant change in flow instability. There exist two kinds of instability; flow undergoes a primary instability (Hopf bifurcation) at a low Reynolds number, and the resulting time-periodic two-dimensional flow subsequently becomes unstable to three-dimensional disturbances at a higher Reynolds number (secondary instability). We report here the dependencies of the primary instability as well as the flow characteristics of the subsequent time-periodic 2D flow, including flow-induced forces and Strouhal number of vortex shedding, on the distance between the cylinders and the channel wall. We also present a Floquet stability analysis on the time-periodic 2D flows to identify the onset of the secondary instability leading to 3D flow.

[1]This work was supported by UVRC, Korea.

LA.00051 Non-linear stability and transition analysis in reactive hypersonic shear-layers , LUCA MASSA, University of Texas at Arlington — Carbon chemistry and the endothermic reactions it supports were previously shown to delay hypersonic boundary layer instability and transition. The present analysis addresses the analogous problem in free shear-layers and arrives to the conclusion that the lack of the acoustic trapping mechanism implies that endothermic chemistry can lead to stabilization or destabilization of the shear-layer depending on the free-stream temperature. This study identifies three mechanism by which carbon chemistry affects instability and transition. The first is rooted in the changes to the inflectional profiles caused by the visco- chemical interaction. The second is due to damping of the perturbation by finite rate chemistry. The third is linked to streamwise relaxation which delays the onset of secondary instability of vortical structures generated by a saturated primary instability wave. Linear analysis predicts changes in growth rate lower than 30% for Mach numbers below 5. Nonlinear parabolized stability analysis predicts significantly larger differences, depending on whether the primary or secondary instability trigger the transition onset.

LA.00052 Dispersion of capillary waves in elliptical cylindrical jets , GHOBAD AMINI, ALI DOLATABADI, Concordia University — In this work motion of a low speed liquid jet issuing from an elliptic orifice through the air is studied. Mathematical solution of viscous free-surface flow for this asymmetric geometry is simplified by using one-dimensional Cosserat (directed curve) equations which can be assumed as a low order form of Navier-Stokes equations for slender jets. Linear solution is performed and temporal and spatial dispersion equations are derived. Growth rate and phase speed of unstable and stable modes under various conditions are presented. The possibility of instability of asymmetric disturbances is studied too. With distance down the jet, major and minor axes are altered and finally jet breaks up due to capillary instability. The effect of jet velocity and viscosity and also orifice ellipticity on axis-switching and breakup is investigated.

LA.00053 Influence of confinement by smooth and rough walls on particle dynamics in dense hard-sphere suspensions , BURAK ERAL, DIRK VAN DEN ENDE, MICHEL DUITS, FRIEDER MUGELE, Physics of Complex Fluids Group, University of Twente, PHYSICS OF COMPLEX FLUIDS GROUP, UNIVERSITY OF TWENTE TEAM — We used video microscopy and particle tracking to study the dynamics of confined hard-sphere suspensions. Our fluids consisted of 1.1-μm-diameter silica spheres suspended at volume fractions of 0.33–0.42 in water-dimethyl sulfoxide. Suspensions were confined in a quasiparallel geometry between two glass surfaces. First, as the separation distance (H) is decreased from 18 to 1 particle diameter, a transition takes place from a subdiffusive behavior (as in bulk) at large H, to completely caged particle dynamics at small H. These changes are accompanied by a strong decrease in the amplitude of the mean-square displacement (MSD) in the horizontal plane parallel to the confining surfaces. In contrast, the global volume fraction essentially remains constant when H is decreased. Second, measuring the MSD as a function of distance from the confining walls, we found that the MSD is not spatially uniform but smaller close to the walls.. Although confinement also induces local variations in volume fraction, the spatial variations in MSD can be attributed only partially to this effect. The changes in MSD are predominantly a direct effect of the confining surfaces. Hence, both the wall roughness and the separation distance (H) influence the dynamics in confined geometries.

LA.00054 Large-scale structures of the turbulent boundary layer in wall-normal/spanwise plane[1]

, JAE HWA LEE, JIN LEE, HYUNG JIN SUNG, KAIST — Coherent wall-normal structures in turbulent boundary layer (TBL) have been investigated by scrutinizing the direct numerical simulation (DNS) dataset with Re_θ=2500. The spatial signatures of hairpin vortex legs were frequently observed in the vector fields of the wall-normal/spanwise (yz) plane and it was found that groups of such hairpin legs induce wall-normal-aligned elongated structures with a large Reynolds shear stress; this result strongly supports the typical hairpin packet model. The two-point correlation of the velocity fluctuations showed that the wall-normal length scales vary linearly with the distance from the wall and the two-point correlations between the signed swirling motions showed that interactions between counter-rotating vortices are predominant throughout the boundary layer and especially frequent in the near-wall region.

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

LA.00055 Observations of Anisotropy in Atmospheric Turbulence by Means of Moiré Deflectometry

, SAIFOLLAH RASOULI, M.D. NIRY, Y. RAJABI, A.A. PANAHI, Department of Physics, Institute for Advanced Studies in Basic Sciences (IASBS), Zanjan 45137-66731, Iran, J.J. NIEMELA, The Abdus Salam ICTP — We report observations of strong anisotropy in the statistical properties of atmospheric turbulence, using a method based on moiré deflectometry. By combining use of a telescope with moiré deflectometry we achieve a high sensitivity to fluctuations in the wave-front phase, which are, in turn, related to fluctuations in the fluid density. As phase fluctuations of the wave front in the aperature of the telescope are imaged on the first grating of the moiré deflectometer, a high spatial resolution is achieved. Experimentally, we measure covariance of angle of arrival (AA) between pairs of points displaced spatially on the telescope aperature and find significant differences between scaling exponents derived for covariances in the longitunal and transverse directions. We note that the method does not require the use of the Taylor hypothesis and has the advantage of being relatively simple and inexpensive.

LA.00056 Large Eddy Simulation of Turbulent Flow in a Ribbed Pipe[1]

, CHANGWOO KANG, KYUNG-SOO YANG, Inha University, Korea — Turbulent flow in a pipe with periodically wall-mounted ribs has been investigated by large eddy simulation with a dynamic subgrid-scale model. The value of Re considered is 98,000, based on hydraulic diameter and mean bulk velocity. An immersed boundary method was employed to implement the ribs in the computational domain. The spacing of the ribs is the key parameter to produce the d-type, intermediate and k-type roughness flows. The mean velocity profiles and turbulent intensities obtained from the present LES are in good agreement with the experimental measurements currently available. Turbulence statistics, including budgets of the Reynolds stresses, were computed, and analyzed to elucidate turbulence structures, especially around the ribs. In particular, effects of the ribs are identified by comparing the turbulence structures with those of smooth pipe flow. The present investigation is relevant to the erosion/corrosion that often occurs around a protruding roughness in a pipe system.

[1]This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology (2010-0008457).

LA.00057 Simulation of Colliding Vortex Rings and Parallel Performance of Lattice Boltzmann Method

, BAILI ZHANG, MING CHENG, JING LOU, Institute of High Performance Computing — Vortex motion or flow with vorticity plays an essential role in almost all kinds of fluid motion of interest. In order to investigate the mechanisms of vortex reconnection, numerical simulation of collision of two vortex rings in three dimensions was carried out using lattice Boltzmann method. We have studied several runs with different initial angles and positions of the ring planes, i.e., head-on collision, inclined collision and off-side collision, keeping the other parameters fixed. However, in order to achieve high resolution on vortex reconnection and its coherent structures, hundreds million of lattices are often necessary in a three dimensional domain, parallel implementations are adopted to attend the demand of an expressive memory amount and processing power of the method. Two vortex collisions have been used as case study to evaluate our implementation, and very good parallel performance was achieved up to thousands of CPUs in a distributed computer system. The lattice Boltzmann method has proved to be an important technique for the numerical solution of partial differential equations because it has nearly ideal scalability on parallel computers for many applications.

LA.00058 ABSTRACT WITHDRAWN —

LA.00059 Nonlinear pulsations of a Hamiltonian system of the fourth order by a nonlinear trigonometric series

, GEORGE MIROSHNIKOV, The City College of the City University of New York — Dynamics of Hamiltonian systems is the key issue of solitary waves since the initial-value problems on free surfaces and interfaces are reduced to Hamiltonian problems in the reference frame moving with the wave. The Hamiltonian approach covers applications at high Reynolds numbers, which range from the famous irrotational Boussinesq-Rayleigh solitary wave to the rotational waves with a uniform vorticity. The Hamiltonian system with a polynomial potential of the fourth order is studied in the asymmetric case of subcritical periodic pulsations by using a nonlinear trigonometric series in even powers of cosine. The series solutions are computed symbolically and compared with the numerical solution using the Fehlberg fourth-fifth order Runge-Kutta method with degree four interpolant. It is shown that the series solutions with uniform convergence are superior to the numeric solutions with local convergence. The qualitative comparison of the theoretical solutions with the experimental profiles of the Geminga pulsar is also provided.

LA.00060 Thermophoresis of a thermally responsive polymer[1]

, KLINTON KILGORE, JACOB FORD, JENNIFER KREFT PEARCE, The University of Texas at Tyler — Thermophoresis is the migration of a species in response to a temperature gradient. This mechanism can be used to manipulate molecules in microfluidic channels. We experimentally investigate thermophoresis of a synthetic polymer. The polymer is thermally responsive and changes conformation in aqueous solution at the cloud point temperature. Two forms of the polymer are present in the temperature gradient at an average temperature near the cloud point, with one conformation migrating to the hot side of the channel and the other to the cold. We also investigate the polymer's migratory dependence on dissolved ionic species, specifically NaCl, which changes the cloud point temperature. We examine the effect of the conformation change on thermophoresis using a lattice-Boltzmann- based simulation.

[1]Acknowledgment is made to the Donors of the American Chemical Society Petroleum Research Fund for support of this research.

LA.00061 An experimental investigation of the multiphase flows in a photobioreactor for algae cultivation , ZIFENG YANG, Wright State University, HUI HU, MATTEO DEL NINNO, ZHIYOU WEN, Iowa State University — Algal biomass is a promising feedstock for biofuels production, with photobioreactors being one of the major cultivation systems for algal cells. Light absorption, fluid dynamics, and algal metabolism are three key factors in determining the overall performance of a photobioreactor. The behavior of the multiphase flow (i.e., liquid phase – water, gas phase – CO_2 and O_2, and solid phase – algal cells) and turbulent mixing inside the reactor are the core connecting the three factors together. One of the major challenges in the optimal design of photobioreactors for algae cultivation is the lack of in-depth understanding of the characteristics of the multiphase flows and turbulent mixing. In this study, we present a comprehensive experimental study to investigate the effects of turbulent mixing in photobioreactors on the performance of a photobioreactor for algae cultivation. A high-resolution particle image velocity (PIV) system is used to achieve time-resolved, in-situ flow field measurements to quantify the turbulent mixing of the multiphase flows inside the bioreactor, while algal cultures are also grown in the same reactor with the same experimental settings. The mixing characteristics of the multiphase flow are correlated with the algal growth performance in the bioreactors to elucidate the underlying physics to explore/optimize design paradigms for the optimization of photobioreactor designs for algae cultivation.

Monday, November 21, 2011 3:35PM - 5:45PM –

Session L1 Geophysical Flows: Oceanography IV 301

3:35PM L1.00001 Quantifying Parametric Uncertainty in Ocean General Circulation Models: A Sparse Quadrature Approach[1] , JUSTIN WINOKUR, ALEN ALEXANDERIAN, IHAB SRAJ, Johns Hopkins University, MOHAMED ISKANDARANI, ASHWANTH SRINIVASAN, University of Miami, CARLISLE THACKER, NOAA, OMAR KNIO, Johns Hopkins University — We use Polynomial Chaos (PC) expansions to quantify propagation of parametric uncertainties in Ocean General Circulation Models (OGCMs). We focus on short-time, high-resolution simulations in Gulf of Mexico with wind stresses corresponding to hurricane Ivan. A non-intrusive sparse quadrature approach is used to determine the PC coefficients providing a detailed representation of the stochastic model response. The quality of the PC representation is examined through a systematic refinement of the number of resolution levels. The resulting PC representation is then utilized in computing distributions of model variables and analyzing local and global sensitivity of the solution to uncertain parameters.

[1]Work supported by the Office of Naval Research.

3:48PM L1.00002 Large-eddy simulation of large-scale convection cells in unstably stratified open channel flow , ANDRES TEJADA-MARTINEZ, Universty of South Florida, GUILLAUME MARTINAT, Old Dominion University, RACHEL WALKER, University of South Florida, CHESTER GROSCH, Old Dominion University — Results are presented from large-eddy simulation of unstably stratified open channel flow driven by a pressure gradient with zero surface shear stress and a no-slip bottom. Unstable stratification is imposed by a constant cooling flux at the surface and an adiabatic bottom wall. Under neutrally stratified conditions, the flow is characterized by weak full-depth streamwise cells similar to Couette cells in plane Couette flow. Surface cooling leads to stronger full-depth convection cells of larger spanwise scale. Surface cooling increases vertical and spanwise velocity fluctuations in the upper half of the channel, while increasing mixing throughout the water column. Similarities and differences between the flow with full-depth convection cells and a second flow with full-depth Langmuir cells generated via surface wave-current interaction will be highlighted. Comparison between flows is based on visualizations and diagnostics including (i) profiles of mean velocity, (ii) profiles of resolved Reynolds stress components, (iii) invariants of the resolved Reynolds stress anisotropy tensor and (v) balances of the transport equations for mean resolved turbulent kinetic energy.

4:01PM L1.00003 Approximate deconvolution large eddy simulation of a barotropic ocean circulation model , ANNE STAPLES, OMER SAN, Virginia Tech — We investigate a new large eddy simulation closure modeling strategy for two-dimensional turbulent geophysical flows. This closure modeling approach utilizes approximate deconvolution, which is based solely on mathematical approximations and does not employ additional phenomenological arguments. The new approximate deconvolution model is tested in the numerical simulation of the wind-driven circulation in a shallow ocean basin, a standard prototype of more realistic ocean dynamics. The model employs the barotropic vorticity equation driven by a symmetric double-gyre wind forcing, which yields a four-gyre circulation in the time mean. The approximate deconvolution model yields the correct four-gyre circulation structure predicted by a direct numerical simulation, but on a coarser mesh and at a fraction of the computational cost. This first step in the numerical assessment of the new model shows that approximate deconvolution could be a viable tool for under-resolved computations in the large eddy simulation of more realistic turbulent geophysical flows.

4:14PM L1.00004 Mesoscale- and submesoscale-resolving simulations with an anisotropic Smagorinsky subgrid model , SANJIV RAMACHANDRAN, AMIT TANDON, University of Massachusetts Dartmouth, AMALA MAHADEVAN, Woods Hole Oceanographic Institution — The oceanic submesoscales are motions of scale O(100m-1km) that lie between the large, O(100km), mesoscales and the smaller, O(10m), three dimensional eddies. Past studies showed submesoscales are critical to the evolution of fronts in the ocean and to the ecological cycle. This study discusses results from submesoscale-resolving simulations with a nonhydrostatic ocean model employing an anisotropic Smagorinsky subgrid model. Our simulated domain is O(100km) in the horizontal and O(100m) in the vertical with highly anisotropic grid resolutions, O(500-1000m) and O(10m), respectively. The domain resolves both mesoscale and submesoscale eddies. Past studies with similar domains have achieved horizontal subgrid mixing either through constant eddy-viscosities or implicitly through the underlying numerical algorithm. We present results from simulations with and without surface winds. Intense submesoscale activity occurs near the fronts and is associated with O(1) Rossby numbers. With winds, the subgrid dissipation of resolved kinetic energy is strongest in regions with negative potential vorticity and the vertical buoyancy flux is enhanced in the presence of baroclinicity below the mixed layer. Without winds, the subgrid dissi pation and vertical velocity are smaller than in the wind-driven case.

4:27PM L1.00005 Turbulence, mixing, and blooms at ocean fronts , JOHN TAYLOR, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, RAFFAELE FERRARI, Department of Earth, Atmospheric and Planetary Sciences, Massachusetts Institute of Technology — Regions with large horizontal density gradients, or fronts, are ubiquitous features of the upper ocean. As locations where density surfaces outcrop from the ocean interior, fronts serve as conduits for transport of fluid properties, linking the deep ocean and the atmosphere. Although fronts are under-resolved in most global ocean models, recent work has shown that they strongly affect the large-scale circulation and biology of the ocean. This talk will describe results from recent studies based on large-eddy simulations (LES), which find that turbulent mixing is strongly affected by fronts and is subject to two competing effects: turbulence is generated from the available potential energy associated with the front, but vertical mixing is inhibited by the stable stratification that develops as the front slumps. By coupling a simple biological model with the LES, we find that reduced vertical mixing at fronts can trigger phytoplankton blooms in light-limited conditions. These results help explain satellite-based observations of unexpected mid-ocean blooms at high latitudes.

4:40PM L1.00006 A Variable Resolution Global Ocean Model, MARK PETERSEN, Los Alamos National Laboratory, LANL COSIM MPAS-OCEAN DEVELOPMENT TEAM — The Model for Prediction Across Scales (MPAS) is a new software framework for the rapid development of climate model components on unstructured grids. The grids may be quasi-uniform or variable density, on a sphere or rectangular domain, and may use quadrilateral cells, triangle cells, or Voronoi tessellations. MPAS variable density grids are particularly well suited to regional climate simulations. MPAS is developed cooperatively by NCAR MMM and the LANL COSIM team. The MPAS-Ocean component now includes most of the features of a full ocean-climate model. High resolution global simulations with full bathymetry have been run for hundreds of simulated years and produce realistic currents and eddying behavior. MPAS-Ocean may be run with z-level or isopycnal vertical coordinates, and includes high-order horizontal and vertical advection and implicit vertical mixing. Current efforts focus on split barotropic/baroclinic timestepping and improving performance.

4:53PM L1.00007 Evaluation and Improvement of RANS models for Stably Stratified Turbulence, FARID KARIMPOUR, SUBHAS VENAYAGAMOORTHY, Colorado State University — The focus of this study is to account for the effects of buoyancy forces in RANS models for stratified turbulence. To this end, changes to the stratification parameters that account for buoyancy effects in RANS models are proposed. DNS data of stably stratified turbulence are used to study the parameters in two equation turbulence models such as the buoyancy parameter $C_{\epsilon 3}$, and the turbulent Prandtl number Pr_t in the k-ϵ model. Both the gradient Richardson number $Ri = N^2/S^2$, where N is the buoyancy frequency and $S = d\overline{u}/dz$ is the mean shear rate, and the turbulent Froude number $Fr_k = \epsilon/(Nk)$, are used as correlating parameters to characterize stratification in the k-ϵ model. We show that it may be more appropriate to use Fr_k as the parameter of choice for modeling the stratification parameters in the k-ϵ model since it is based on the local properties of the turbulence as opposed to Ri, which is a mean property of the flow. The proposed modifications were implemented in a 1-D water column model called the General Ocean Turbulence Model (GOTM) and used to simulate stably stratified channel flows. The results from numerical simulations using the modified k-ϵ model are compared to DNS data of stably stratified channel flow to assess its efficacy. This modified formulation is also compared with other stability functions in GOTM.

5:06PM L1.00008 Energy Cascade in the Regime of Realistic Ocean Circulation[1], B.T. NADIGA, W.R. CASPER, Los Alamos National Lab — Ocean circulation is forced at the large scales and the instability of the resulting large-scale flow gives rise to intermediate-scale eddies. The large-scale flow and the resultant eddies are both in approximate geostrophic balance; such balance results in an inverse cascade of energy. Consequently, small scale dissipation becomes ineffective and dissipation is limited to interactions of the larger and intermediate scale flow structures with solid boundaries. We consider the modification of this asymptotic behavior in the presence of a range of scales over which unbalanced motions are possible.

[1]This work was carried out under the LDRD-ER program (20110150ER) of the Los Alamos National Laboratory.

5:19PM L1.00009 Examination of the Turbulent Kinetic Energy Budgets in the Mid-Water Column of the Chesapeake Bay Estuary, LUKSA LUZNIK, Mechanical Engineering Department, US Naval Academy , LOUISE WALLENDORF, Hydromechanics Laboratory, US Naval Academy — A local turbulent kinetic energy (TKE) balance is examined from measurements obtained during two-week long field experiment in the Chesapeake Bay near Kent Island, MD, under low to moderate wind conditions. Velocity data were collected with two vertically separated Acoustic Doppler Velocimeters and an upward looking pulse coherent profiler covering 0.8 to 2.5 m above bottom in approximately 5 m of water. Additionally, upward oriented AWAC mounted on a separate frame was used to obtain water column tidal current vertical profiles and overlying wave directional system. Stratification is monitored using two vertically separated CT sondes and estimated Ozmidov scale ranges from very small values up to 5m. Occurrences of large TKE dissipation rates coincide with times of maximum bottom boundary layer shear production with consistently larger values during ebb flows. In general dissipation rate exceeds shear production minus buoyancy flux in the mid-water column. Potential sources of enhanced levels of dissipation are examined including vertical turbulent transport, wind driven shear production and presence of surface gravity waves.

5:32PM L1.00010 ABSTRACT WITHDRAWN —

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L2 Instability in Boundary Layers II 302

3:35PM L2.00001 Transition to Turbulence and Heat-Transfer Overshoot in an Adverse Pressure Gradient High-Speed Boundary Layer[1], KENNETH FRANKO, SANJIVA LELE, Stanford University, STANFORD UNIVERSITY TEAM — Spatial direct numerical simulations (DNS) of transitional high-speed boundary layers with zero and adverse pressure gradients and an isothermal wall are presented for different transition scenarios. The maximum momentum thickness Reynolds numbers vary between 2500 and 5500 and the edge Mach numbers vary between 6 and 4.8 for the different cases presented. Disturbances are introduced into the initially laminar boundary layer through suction and blowing at the wall. Different transition scenarios including first mode oblique breakdown and second mode fundamental resonance are explored. The presence of an adverse pressure gradient accelerates the transition process. Comparisons of the nonlinear modal growth and breakdown for the different initial forcing scenarios will be shown. First mode oblique breakdown is shown to lead to an overshoot in heat transfer and the most rapid development to a turbulent state.

[1]Support provided by AFOSR through STTR to Cascade Technologies

3:48PM L2.00002 ABSTRACT WITHDRAWN —

4:01PM L2.00003 Effect of Surface Curvature on Crossflow Instability in Swept-Wing Boundary Layers, M. MALIK, W. LIAO, F. LI, M. CHOUDHARI, C. CHANG, NASA Langley Research Center — A three-dimensional boundary layer is subject to crossflow instability that manifests itself in the form of stationary or traveling disturbances. Stationary disturbances are induced by surface roughness while free stream turbulence induces traveling disturbances. It is known that convex surface curvature tends to stabilize crossflow instability while the mean flow non-parallel effect is generally destabilizing, with the net effect being mildly stabilizing when compared to the results obtained using quasi-parallel linear stability theory. Here, an analysis is performed for two swept airfoils using parabolized stability equations that account for both the surface curvature and the non-parallel effect. One airfoil has larger convex curvature than the other, where the convex surface curvature is scaled by defining a Gortler number. The net decrease in the stationary crossflow N factor is about 6 for the airfoil with stronger curvature. The analysis suggests that, if transition is induced by stationary crossflow disturbances, then surface curvature could be used as a control parameter for natural laminar flow design. The strong effect of surface curvature on stationary disturbances highlights the importance of investigating the receptivity of stationary and traveling disturbances since the latter are much less influenced by surface curvature resulting in much higher relative N factors for traveling disturbances.

4:14PM L2.00004 Secondary instability analysis of pre-transitional streaks in boundary layers

, M.J. PHILIPP HACK, TAMER ZAKI, Imperial College London — In the presence of free-stream vortical disturbances, laminar boundary layers develop streamwise-elongated perturbations of high amplitude, commonly known as Klebanoff streaks. The regions of shear surrounding these primary structures provide the potential for the growth of secondary instabilities which ultimately initiate bypass transition. By means of linear analysis, we examine the secondary instability which precedes the formation of turbulent spots. The base state is extracted from direct numerical simulations of the bypass process. The simulation setup is similar to the work of Jacobs & Durbin (2001), where transition is triggered by broadband free-stream vortical forcing. The velocity field therefore includes a spectrum of streaks with different structures and amplitudes. The stability analysis can nevertheless identify the streaks which indeed develop secondary instabilities and break down to turbulence. The predictions of linear theory, in particular the instability wavelength and phase speed, are compared to the streak instabilities recorded in the DNS of the full bypass process.

4:27PM L2.00005 Conditional Sampling of Bypass Transition in Pressure Gradient Boundary Layers

, KEVIN NOLAN, TAMER ZAKI, Imperial College London — Conditional sampling of velocity fields from Direct Numerical Simulations (DNS) of bypass transition is performed with discrimination between laminar and turbulent events. Individual positive and negative streaks are isolated and an extreme value analysis of their amplitudes is performed. A more detailed view of the growth of positive and negative streaks is obtained than is typical by simply measuring the root mean square perturbations. The resulting velocity distributions are compared with the amplitudes of streaks which undergo secondary instability, and breakdown into turbulent spots. A range of pressure gradients is considered and the rates of turbulent spot production and propagation are investigated. While the spot production rate increases significantly with adverse pressure gradient, it is found that the spot propagation rate is unaffected. By considering the spot spreading angle others have shown a pressure gradient dependence of the propagation parameter. The current work eschews the spreading angle, opting instead to directly track the growth of the spot volume.

4:40PM L2.00006 Attachment-Line Heating in a Compressible Flow

, HELEN REED, WILLIAM SARIC, Texas A&M University — The attachment-line boundary layer on a swept wing can be subject to either an instability or contamination by wing-root turbulence. A model of the attachment-line boundary layer is first developed including compressibility and wall heating in a Falkner-Skan-Cooke class of 3-D boundary layers with Hartree parameter of 1.0. For cases otherwise subcritical to either contamination or instability, the destabilizing effect of leading-edge heating under a variety of sweep angles and flight conditions is demonstrated. The results correlate with the attachment-line Reynolds number. Because the required heating levels are reasonable and achievable to trip the flow over the wing to turbulent, one possible application of this work is in the establishing of a baseline turbulent flow (on demand) for the calibration of a laminar-flow-control health monitoring system. **Portion based on work under Framework Agreement between Airbus Americas and NIA, and opinions, findings, conclusions do not necessarily reflect views of Airbus or NIA. **Support from AFOSR/NASA National Center for Hypersonic Research in Laminar-Turbulent Transition through Grant FA9550-09-1-0341 gratefully acknowledged.

4:53PM L2.00007 Transition prediction for oblique breakdown in supersonic boundary layers with uncertain disturbance spectrum

, GENNARO SERINO, OLAF MARXEN, PATRICK RAMBAUD, THIERRY MAGIN, von Karman Institute for Fluid Dynamics — Prediction of laminar-turbulent transition is important for the design of heat shields for planetary (re)-entry vehicles. The heat load may increase significantly if a previously laminar boundary layer on the vehicle surface becomes turbulent. Transition-prediction methods based on linear stability theory, such as the e^N-method, offer an attractive compromise between simplicity and accuracy. However, non-linear stages of disturbance evolution as well as the receptivity stage are neglected, hampering the general use of these methods. Here we perform an investigation of the oblique breakdown scenario. In this scenario, a pair of oblique waves is convectively amplified and quickly leads to turbulence once these waves reach an amplitude of approximately two percent. This knowledge allows us to define a simple breakdown criterion as a model for the non-linear stage. The receptivity process, whose outcome provides the initial disturbance amplitudes, may not be as easily modeled. Flow physics of the receptivity process are neglected here. Instead, we assume an uncertain disturbance spectrum, which depends on the disturbance frequency and spanwise wave number. Using stochastic collocation, linear stability theory is then employed to yield a probabilistic transition prediction.

5:06PM L2.00008 ABSTRACT WITHDRAWN —

5:19PM L2.00009 The pre-cursor of Kelvin-Helmholtz instabilities in wake-perturbed separated boundary layers[1]

, AYSE G. GUNGOR, MARK P. SIMENS, JAVIER JIMÉNEZ, U. Politécnica Madrid — The interaction of large-scale wake disturbances with a pressure-induced separation bubble on a flat plate is studied by direct numerical simulation. The space-time development of the separated region shows roll-up vortices in the separated shear layer. Their appearance is closely associated with the receptivity of the upstream attached boundary layer. The wake-passing excites a linear normal mode of the boundary layer. This mode, which is initially very small, appears in the form of two-dimensional wave-trains upstream of the separation point, but becomes unstable and transforms into the Kelvin-Helmholtz mode as the profile separates. Studies based on varying the frequency or modifying the shape of the forcing further support this scenario of the initial development of the rolls in the separated shear layer, and show that the influence of the wakes is not to directly force the Kelvin-Helmholtz instability of the separation bubble, but to induce perturbations upstream in the attached boundary that eventually seed the instability of the separated shear layer.

[1]Funded by CICYT and CONSOLIDER.

5:32PM L2.00010 Stability of unsteady flow in a rotating torus

, RICHARD HEWITT, ANDREW HAZEL, University of Manchester, RICHARD CLARKE, University of Auckland, JAMES DENIER, University of Adelaide — We consider the temporal evolution of a viscous incompressible fluid in a torus of finite curvature; a problem first investigated experimentally by Madden and Mullin (1994), herein referred to as MM. The system is initially in a state of rigid-body rotation (about the axis of rotational symmetry) and the container's rotation rate is then changed impulsively. We describe the transient flow that is induced at small values of the Ekman number, over a time scale that is comparable to one complete rotation of the container. We show that (rotationally symmetric) eruptive singularities (of the boundary layer) occur at the inner or outer bend of the pipe for a decrease or an increase in rotation rate respectively. Moreover, there is a ratio of initial-to-final rotation frequencies for which eruptive singularities can occur at both the inner and outer bend simultaneously. We also demonstrate that the flow is susceptible to non-axisymmetric inflectional instabilities. The inflectional instability arises as a consequence of the developing eruption and is shown to be in qualitative agreement with the experimental observations of MM. Detailed quantitative comparisons are made between asymptotic predictions and finite (but small) Ekman number Navier-Stokes computations using a finite-element method.

Monday, November 21, 2011 3:35PM - 5:45PM —

Session L3 Dynamical Systems and Chaos II 303

3:35PM L3.00001 Burning invariant manifolds for propagating fronts in a chain of vortices[1] , TOM SOLOMON, Bucknell University, MARK KINGSBURY, Bucknell University, JOHN MAHONEY, KEVIN MITCHELL, University of California at Merced — We present experimental studies of the behavior of reaction fronts in a chain of alternating vortices. The flow is produced by a magnetohydrodynamic forcing technique, and the fronts are produced by the ferroin-catalyzed Belousov-Zhabotinsky chemical reaction. We introduce *burning invariant manifolds* (BIMs) which act as barriers to front propagation, similar to the role played by invariant manifolds as barriers to passive transport in two-dimensional flows. Unlike manifolds for passive transport, though, BIMs are one-sided barriers, passing either left- or right-going fronts but blocking the other. We show how the BIMs can be measured experimentally for both time-independent and time-periodic flows. The experimental results are compared to simulations based on a simplified numerical model of the flow.

[1]Supported by NSF Grants DMR-0703635, DMR-1004744, PHY-0552790 and PHY-0748828.

3:48PM L3.00002 Lobe dynamics and front propagation in advection-reaction-diffusion systems[1] , JOHN MAHONEY, KEVIN MITCHELL, University of California, Merced, TOM SOLOMON, Bucknell University — We consider the addition of reaction-diffusion dynamics to systems undergoing chaotic advection. This can be viewed as a simplified model of diverse systems such as combustion dynamics in a chaotic flow, microfluidic chemical reactors, and blooms of phytoplankton and algae. Recently, we have proposed that front propagation in these systems is strongly influenced by burning invariant manifolds (BIMs)—geometric structures analogous to traditional invariant manifolds for passive transport. Additionally, BIMs may be used to define tangle-like structures that support a version of lobe dynamics for front propagation. In this talk, we discuss the theory and structure of BIMs and demonstrate the modified lobe-dynamics. We also present a potential application of the lobe dynamics to the control of reactive flows.

[1]Supported by NSF Grants DMR-0703635, DMR-1004744, PHY- 0552790 and PHY-0748828.

4:01PM L3.00003 The geometry of mode-locked fronts in periodically driven advection-reaction-diffusion systems[1] , KEVIN MITCHELL, JOHN MAHONEY, University of California, Merced, TOM SOLOMON, Bucknell University — We consider reaction-diffusion dynamics within a periodically driven fluid forming a linear chain of alternating vortices. Prior theoretical and experimental work on this system has demonstrated rich structure in the dynamics of the reaction front; for certain parameter values, the front will be mode locked to the external driving. Using dynamical systems theory, we relate the mode-locking behavior to the existence of relative periodic orbits (RPOs), and we show that the mode-locked front itself follows the profile of a "burning invariant manifold" (BIM)—a generalization of traditional invariant manifolds for passive transport, which incorporates the dynamics of front propagation. Together, the RPOs and BIMs provide clear criteria for the emergence and destruction of mode locking as well as an explanation of the front geometry.

[1]Supported by NSF Grants DMR-0703635, DMR-1004744, PHY- 0552790 and PHY-0748828.

4:14PM L3.00004 Barriers to reaction front propagation in a spatially random, time-independent flow[1] , DYLAN BARGTEIL[2], Bucknell University, TOM SOLOMON, Bucknell University, JOHN MAHONEY, KEVIN MITCHELL, University of California at Merced — We present experimental studies of barriers, called *burning invariant manifolds* (BIMs), to front propagation in a spatially random, time-independent flow. We generate the flow with a magnetohydrodynamic technique that uses a DC current and a disordered pattern of permanent magnets. The velocity field is determined from this flow using particle tracking velocimetry, and reaction fronts are produced using the Ferroin-catalyzed Belousov-Zhabotinsky (BZ) chemical reaction. We use the experimental velocity field and a three-dimensional set of ODEs to predict from theory the location and orientation of BIMs. These predicted BIMs are found to match up well with the propagation barriers observed experimentally in the same flow using the BZ reaction. We explore the nature of BIMs as one-sided barriers, in contrast to invariant manifolds that act as barriers for passive transport in all directions. We also explore the role of projection singularities in the theory and how these singularities affect front behavior.

[1]Supported by NSF Grants DMR-0703635, DMR-1004744, PHY-0552790 and PHY-0748828.
[2]Current address: University of Maryland at College Park

4:27PM L3.00005 Computation of Lagrangian Coherent Structures from their Variational Theory , MOHAMMAD FARAZMAND, Mathematics and Statistics, McGill University, Canada, MANIKANDAN MATHUR, Laboratoire de Meteorologie Dynamique, Ecole Polytechnique, 91128 Palaiseau, France, GEORGE HALLER, Mechanical Engineering, McGill University, Canada — We describe a computational algorithm for detecting hyperbolic Lagrangian Coherent Structures (LCS) from a recently developed variational theory [1]. In contrast to earlier approaches to LCS, our algorithm is based on exact mathematical theorems that render LCS as smooth parametrized curves, i.e., trajectories of an associated ordinary differential equation. The algorithm also filters out LCS candidates that are pure artifacts of high shear. We demonstrate the algorithm on two-dimensional flow models and on an experimentally measured turbulent velocity field.

[1] G. Haller, A variational theory of hyperbolic Lagrangian Coherent Structures, *Physica D* **240** (2011) 574-598

4:40PM L3.00006 Geodesic Theory of Transport Barriers in Unsteady Flows , GEORGE HALLER, McGill University, FRANCESCO BERON-VERA, University of Miami — We introduce a unified approach to detecting finite-time Lagrangian transport barriers in two-dimensional unsteady flows with general time dependence. Seeking transport barriers as least deforming material lines, we obtain a variational formulation for such barriers. This variational problem turns out to be well-posed only for three types of transport barriers: hyperbolic barriers (generalized stable and unstable manifolds), elliptic barriers (generalized KAM curves or eddy boundaries), and parabolic barriers (generalized shear jets). Such barriers then coincide with minimal geodesics under an appropriate metric induced by the Cauchy-Green strain tensor on the initial configuration of the flow. The geodesics are obtained as a solution of an ordinary differential equation, and hence are available in a smooth, parametrized form. We show how these new results reveal previously unknown transport barriers in complex model flows and geophysical data sets.

4:53PM L3.00007 Noise induced state transitions, intermittency and universality in the noisy Kuramoto-Sivashinsky equation , MARC PRADAS, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK, GREG PAVLIOTIS, DEMETRIOS PAPAGEORGIOU, Department of Mathematics, Imperial College London, London SW7 2AZ, UK, DMITRI TSELUIKO, School of Mathematics, Loughborough University, Leicestershire LE11 3TU, UK — The Kuramoto-Sivashinsky (KS) equation is a paradigmatic model for a wide spectrum of systems exhibiting spatio-temporal chaos, such as a thin-liquid film falling down a vertical substrate. Here we deal with the noisy KS eqiation which can be derived for the falling film problem with a topographically random substrate. We examine the effects of additive noise in a regime close to the instability onset. We show that when the noise is highly degenerate, in the sense that it acts only on the first stable mode, the solution of the KS equation undergoes several transitions between different states, including a critical on-off intermittent state that is eventually stabilized as the noise strength is increased. Similar results are obtained with the Burgers equation, which has often been used as a prototype of one-dimensional turbulence. Such noise-induced transitions can be completely characterized through critical exponents, obtaining that both equations belong to the same universality class. The results of our numerical investigations are explained rigorously using multiscale techniques.

5:06PM L3.00008 A new mode reduction strategy applied to the generalized Kuramoto-Sivashinsky equation , MARKUS SCHMUCK, MARC PRADAS, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK, GREG PAVLIOTIS, Department of Mathematics, Imperial College London, London SW7 2AZ, UK, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK — The generalized Kuramoto-Sivashinsky (gKS) equation is one of the simplest prototypes modeling nonlinear active media with energy supply, energy dissipation and dispersion. Not surprisingly, it has been reported for a wide variety of physical settings, from reaction-diffusion systems, e.g. propagation of concentration waves and flame-front instabilities, to fluid dynamics, e.g. a viscous film flowing down an inclined plane. We undertake a combined theoretical-numerical study of the gKS equation. We first approximate it with a renormalization group equation. This approximation forms the basis for a non-standard stochastic mode reduction that guarantees optimality in the sense of maximal information entropy. Herewith, noise is rigorously defined in the reduced gKS equation and hence provides an analytical explanation for its origin. These derivations allow us to develop reliable numerical approximations to the gKS equation and a rigorous methodology on how to add noise. Interestingly, noise becomes increasingly important by decreasing the degrees of freedom in the discretization strategy.

5:19PM L3.00009 Torus Streamlines in the 3D Steady Lid-driven Cavity Flows , KATSUYA ISHII, ITC, Nagoya University, CHIKARA OTA, CES, Nagoya University, SHIZUKO ADACHI, Tokyo International University — Streamlines of the incompressible vortical flows in three-dimensional rectangular cavities with different aspect ratios are numerically studied for several Reynolds numbers by using a combined compact finite difference (CCD) scheme with high accuracy and high resolution. The flow is driven by a lid moving tangentially with constant speed. Non-dimensional geometrical parameters of the cavity are the depth-to-width aspect ratio Γ and the span-to-width aspect ratio Λ. The flow parameter is the Reynolds number Re. We study the flow structures in the square cavity (Γ=1) with the spanwise aspect ratios Λ=1 and 6.55 for Re from 100 to 400. Torus streamlines are obtained from the velocity field of the steady incompressible flow. Several other streamlines show chaotic behavior. They are equivalent to a non-autonomous Hamiltonian system of one-degree-of-freedom. In order to examine the features of the flow pattern with different parameters, we analyze the Poincare sections in the cross sections of cavities.

5:32PM L3.00010 Topological chaos in a lid driven cavity flow at finite Reynolds number , PRADEEP RAO, Virginia Tech, ANDREW DUGGLEBY, Texas A&M, MARK STREMLER, Virginia Tech — Topological chaos, or chaos that is guaranteed to exist in a system due to sufficiently complex motion of a few periodic orbits, has been demonstrated for creeping flow in a lid driven cavity. Nearly-periodic systems can by analyzed in a similar way based on the presence of Almost Invariant Sets (AIS) with similarly complex space-time trajectories. We extend this analysis to finite Reynolds number flows in a lid driven cavity using a 2D Fourier-Chebyshev spectral algorithm for the streamfunction-vorticity formulation, which enables accurate particle tracking that can resolve the exponential stretching of material lines in the flow. Simply extending the Stokes' flow parameters to stirring at finite Reynolds number leads to a decrease in system performance, but tuning the system based on the topological analysis can lead to enhanced stirring.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L4 Drops VII: Impact and Interactions 307

3:35PM L4.00001 Particle-drop Impact in Midair[1] , ALIDAD AMIRFAZLI, Department of Mechanical Engineering, University of Alberta, STEFAN STRZEBIN, OLIVER PEISE, DAVID CHEVROLLIER — For the first time the impact of a drop and particle in the mid-air is studied, which is a fundamental physical phenomenon relevant to many applications involving a fluidized bed and a liquid jet, e.g. drug particle coating, and upgrading of heavy oil. To date it has not been clear what happens when a particle and drop collide in midair. An apparatus was build to allow deterministic impact of a particle and drop to occur in midair. Using high speed imaging the impact of three different particles with water drops is studied at different relative velocities. Possible collision outcomes are elucidated in terms of particle-drop diameter ratio, Weber number, and particle wettability. Three distinct regimes of bonding, ripping and coating, and shattering are identified and discussed in this novel study. The differences between on-axis versus off-axis impact is also briefly discussed.

[1]NSERC & Canada Research Chairs.

3:48PM L4.00002 Gravitational Collisions of Spherical Drops at Finite Stokes Numbers and Low Reynolds Numbers , MICHAEL ROTHER, University of Minnesota Duluth, ROBERT H. DAVIS, University of Colorado at Boulder — Collision efficiencies are calculated by a trajectory analysis for two sedimenting spherical drops with exact methods for determining the hydrodynamic forces at finite Stokes number and low Reynolds number. When the Reynolds number is small, fluid inertia is negligible, and the hydrodynamic forces are linear functions of the translational velocities of the drops. However, at nonzero Stokes numbers, drop inertia must be taken into account, and the hydrodynamic forces do not balance the applied forces. For drops in close approach, lubrication forces and attractive molecular forces are considered. Comparison is made between the effects of unretarded and retarded van der Waals forces and Maxwell slip on collision efficiencies. An important application is to raindrop growth for drop radii between 10 and 30 μm. The collision efficiency goes through a minimum and then approaches the Smoluchowski limit of no hydrodynamic interactions as the drop size and Stokes number become increasingly large. Theoretical predictions are required in this range of drop sizes because experiments are difficult.

4:01PM L4.00003 Bouncing Jets , NAVISH WADHWA, Department of Engineering Science and Mechanics, Virginia Tech, PAVLOS VLACHOS, Department of Mechanical Engineering, Virginia Tech, SUNGHWAN JUNG, Department of Engineering Science and Mechanics, Virginia Tech — Contrary to common intuition, free jets of fluid can "bounce" off each other on collision in mid-air, through the effect of a lubricating air film that separates the jets. We have developed a simple experimental setup to stably demonstrate and study the non-coalescence of jets on collision. We present the results of an experimental investigation of oblique collision between two silicone oil jets, supported by a simple analytical explanation. Our focus is on elucidating the role of various physical forces at play such as viscous stresses, capillary force and inertia. A parametric study conducted by varying the nozzle diameter, jet velocity, angle of inclination and fluid viscosity reveals the scaling laws for the quantities involved such as contact time. We observed a transition from bouncing to coalescence with an increase in jet velocity and inclination angle. We propose that a balance between the contact time of jets and the time required for drainage of the trapped air film can provide a criterion for transition from non-coalescence to coalescence.

4:14PM L4.00004 Bubble entrapment by droplet-meniscus collision , KOEN G. WINKELS, DIEDERIK L. KEIJ, JACCO H. SNOEIJER, Physics of Fluids, University of Twente, The Netherlands — The impact of a sessile droplet with a meniscus, close to a moving contact line, is studied experimentally with high-speed imaging. Above a certain velocity and impacting droplet size, bubbles are entrained into the liquid during the process of coalescence. By looking through the liquid we can resolve the formation of an air sheet that is trapped between the drop and the meniscus. This sheet breaks up into a single or multiple bubbles, depending on the experimental conditions. We characterize the various mechanisms for entrainment and identify scaling relations for the size of the entrained bubbles in terms of impacting droplet size and velocity.

4:27PM L4.00005 ABSTRACT WITHDRAWN –

4:40PM L4.00006 ABSTRACT WITHDRAWN –

4:53PM L4.00007 The Effect of Gas Properties on Mesler Bubble Entrainment , R.K. SUNDBERG, J.R. SAYLOR, Clemson University — Mesler entrainment involves the generation of hundreds of micron size bubbles, frequently distributed in a chandelier- like pattern following the impact of a liquid drop with a bulk surface of the same fluid. To date, research on Mesler entrainment has taken place in air at atmospheric pressure and has therefore neglected to test the influence of gas properties. The results of drop impact studies are presented where a controlled environment was employed consisting of air- helium and air-carbon dioxide mixtures. The dynamic viscosity of pure air at STP is 1.85×10^{-5} Pa.s and the kinematic viscosity is 1.6×10^{-5} m^2/s. By using air-helium and air-carbon dioxide mixtures, kinematic viscosities ranging from 0.5 to 7 times that of air were attained, with dynamic viscosities ranging 0.8 to 1.0 times that of air. The frequency of occurrence of Mesler entrainment is noted for methanol in these environments and is compared with results gathered in air. The effect of the gas phase viscosities and the Capillary number are discussed.

5:06PM L4.00008 Jetting from impact of a spherical drop with a deep layer , LI ZHANG, JAMESON TOOLE, Department of Physics & Center for the Study of Complex Systems, University of Michigan, Ann Arbor, MI 48109 USA, KAMEL FAZZAA, X-Ray Science Division, Argonne National Laboratory, Argonne, IL 60439, USA, ROBERT DEEGAN, Department of Physics & Center for the Study of Complex Systems, University of Michigan, Ann Arbor, MI 48109 USA, DEEGAN GROUP TEAM, X-RAY SCIENCE DIVISION, ADVANCED PHOTON SOURCE COLLABORATION — We performed an experimental study of jets during the impact of a spherical drop with a deep layer of same liquid. Using high speed optical and X-ray imaging, we observe two types of jets: the so-called ejecta sheet which emerges almost immediately after impact and the lamella which emerges later. For high Reynolds number the two jets are distinct, while for low Reynolds number the two jets combine into a single continuous jet. We also measured the emergence time, speed, and position of the ejecta sheet and found simple scaling relations for these quantities.

5:19PM L4.00009 "Effervescent" Atomization in two dimensions , HENRI LHUISSIER, EMMANUEL VILLERMAUX, IRPHE, Aix Marseille Universite — A planar *Savart* water sheet uniformly seeded with small air bubbles in large surface concentration is studied as a model experiment of the so-called "effervescent" atomization process. This two-dimensional setup allows for a quantitative observation of all the steps of the sheet disintegration into a collection of disjointed droplets. The bubbles are heterogeneous nucleation sites which puncture the sheet forming growing holes. The dynamics of the holes opening competes with the simultaneous nucleation rate of new holes in a statistically stationary fashion. The liquid constitutive of the sheet is then transitorily concentrated into a web of ligaments of various lengths and diameters, at the junction between adjacent holes. Their break-up produces the final spray. We provide a complete description of the ligaments web statistics in the case where nucleation is synchronous, and show that the drop size dispersion from the breakup of a single ligament is responsible for the shape of the resulting overall spray drop size distribution.

5:32PM L4.00010 Dynamics of a drop trapped inside a circular hydraulic jump , LUC LEBON, Université Paris 7 / CNRS, ALEXIS DUSCHENE, Université Paris 7, LAURENT LIMAT, Université Paris 7 / CNRS — We investigate the dynamics of a drop trapped inside a circular hydraulic jump : in our experiment, a circular hydraulic jump is formed by a viscous jet impacting a horizontal glass disk. A drop of the same liquid, deposited in the jump does not coalesce, and remains trapped at its periphery, because of the air entrainment linked to the high drop rotation speed. Depending on the flow rate, the drop can exhibit complex dynamics, from regular rotation along the jump to "chaotic" behaviour. We also studied in detail hydrodynamics of the liquid drop and its interaction with the jump. Our investigations show that hydrodynamics of the jump is still not fixed.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L5 CFD V: Numerical Methods II 308

3:35PM L5.00001 On the accuracy of difference formulas at block interfaces in Structured Adaptive Mesh Refinement (S-AMR) methods[1] , MARCOS VANELLA, ELIAS BALARAS, The George Washington University — S-AMR is a cost/efficient strategy to solve a variety of complex problems in laminar and turbulent incompressible flows. The computational grid consists of a number of nested grid blocks at different resolution levels. The coarsest grid blocks always cover the entire computational domain, and local refinement is achieved by the bisection of selected blocks in every coordinate direction. This generates coarse-fine interfaces between blocks whose treatment affects the accuracy and conservation properties of solver. In the present study we will focus on the effects of different interpolation strategies on the properties of finite-differences taken at block boundaries or cells neighboring the boundaries. Looking into a set of model problems we found that for a variety of popular interpolation schemes (both low and high order) the symmetry properties of central difference operators are lost and dissipation and phase errors are added to the errors of the difference formulas. Recurring interpolation, in particular, degrades the quality of formulas as more coarse and fine point values are introduced in an asymmetric manner. Solution strategies based on the minimization of the modified wavenumber error will be presented.

[1]Supported by the NSF

3:48PM L5.00002 A Method of Adaptive Mesh Refinement on Cartesian Unstructured Meshes , CARLOS BALLESTEROS, MARCUS HERRMANN, Arizona State University — A general method for a dynamic meshing library with built-in localized adaptive mesh refinement (AMR) routines for hexahedral meshes is presented. Current block-based AMR methods apply stacked nested finer grids to regions of the computational domain, solve the governing equations of the flow within the fine grid, and then interpolate the fine grid values to the coarse grid. Unlike existing libraries, the proposed method is designed to generate fully unstructured grids that may be refined and coarsened on-the-fly up to an arbitrary level. This approach allows for *h*-refinement without the memory and computational expense of calculating masked coarse grid cells, as is done in block-based AMR. The hexahedral nature of the meshes simplifies the finite-volume algorithms, reducing the computational expense of refinement when compared with arbitrary tetrahedral-mesh solvers. Complex surfaces can be quickly modeled by successive refinement of control volumes while retaining numerical stability and minimizing the size of the grid.

4:01PM L5.00003 A fully conservative finite-volume method for incompressible Navier-Stokes equations on locally refined nested Cartesian grids[1] , ADAMANDIOS SIFOUNAKIS, DONGHYUN YOU, SATBIR SINGH, Carnegie Mellon University — A second-order-accurate finite-volume method is developed for the solution of incompressible Navier-Stokes equations on locally refined nested Cartesian grids. Numerical accuracy and stability on locally refined nested Cartesian grids are achieved using a finite-volume discretization of the incompressible Navier-Stokes equations based on higher- order conservation principles - *i.e.*, in addition to mass and momentum conservation, kinetic energy conservation in the inviscid limit is used to guide the selection of the discrete operators and solution algorithm. Hanging nodes at the interface are *implicitly* slanted to improve the pressure-velocity projection, while the other parts of the grid maintain an orthogonal Cartesian grid topology. The present method is found to significantly improve the computational efficiency while it is straightforward to implement. The present method shows superior conservation of mass, momentum, and kinetic energy compared to the conventional methods employing interpolation at the interface between coarse and fine grids in simulations of Taylor vortex, lid-driven cavity flow, and flow over a square cylinder.

[1]Supported by the Army Research Office Grant W911NF1010348

4:14PM L5.00004 A Space/Time Dynamically Adaptive Method for Multiscale Problems , TEMISTOCLE GRENGA, ZACHARY ZIKOSKI, SAMUEL PAOLUCCI, University of Notre Dame, MAURO VALORANI, "La Sapienza" University of Rome — Systems of partial differential equations (PDEs) describing problems that are multiscale in space and time are computationally very expensive to solve. In order to overcome the challenges related to both thin spatial layers and temporal stiffness we propose the use of a wavelet adaptive multilevel representation (WAMR) in space and an adaptive model reduction method (G-Scheme) in time. The multilevel structure of the algorithm provides a simple way to adapt computational refinements to local demands of the solution. High resolution computations are performed only in spatial regions where sharp transitions occur, while the G-Scheme is an explicit solver developed for stiff problems which is built upon a local decomposition of the dynamics in three subspaces involving slow, active and fast time scales. Only the modes in the active subspace are integrated numerically, the others are approximated asymptotically. Subsequently, the original problem not only becomes substantially smaller, but more importantly non-stiff. Combining the WAMR technique with the G-Scheme yields a time accurate solution of a prescribed accuracy with a much smaller number of space- time degrees of freedom. While the computational scheme can be used to solve a wide class of stiff PDE problems, we will illustrate its use in the solution of the Navier Stokes equations in reactive flows.

4:27PM L5.00005 Parallel Adaptive Wavelet Collocation Method for PDEs[1] , OLEG V. VASILYEV, ALIREZA NEJADMALAYERI, ALEXEI VEZOLAINEN, University of Colorado at Boulder — A parallel adaptive wavelet collocation method for solving a large class of Partial Differential Equations is presented. The parallelization is achieved by developing an asynchronous parallel wavelet transform, which allows to perform wavelet transform and derivative calculations on each processor without additional data synchronization on each level of resolution. The data are stored using tree-like structure with tree roots starting at sufficiently large level of resolution to shorten tree traversing path and to minimize the size of trees for data migration. Both static and dynamic domain partitioning approaches are developed. For the dynamic domain partitioning, trees are considered to be the minimum quantum of data to be migrated between the processors. This allows fully automated and efficient handling of non-simply connected partitioning of a computational domain. Dynamic load balancing is achieved via domain repartitioning during grid adaptation step and reassigning tree data structure nodes to the appropriate processors to ensure approximately the same number of nodes on each processor. The parallel efficiency of the approach is discussed based on parallel Coherent Vortex Simulations of homogenous turbulence with linear forcing at effective non-adaptive resolutions up to 2048^3 using as many as 1024 CPU cores.

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

4:40PM L5.00006 A Hybrid Grid Compressible Flow Solver for Large-Scale Supersonic Jet Noise Simulations on Multi-GPU Clusters[1] , ANDREW CORRIGAN, K. KAILASANATH, JUNHUI LIU, RAVI RAMAMURTI, DOUGLAS SCHWER, Naval Research Laboratory, NAVAL RESEARCH LABORATORY TEAM — A compressible flow solver for multi-GPU clusters has been developed for performing large-scale supersonic jet noise and other high-speed compressible flow simulations over hybrid grids. While supersonic jet noise simulations require the accurate representation of complex nozzle geometry and thus the use of unstructured grids, much of the domain geometry can be represented sufficiently with structured grids, which drastically reduces memory bandwidth consumption and storage. Therefore, hybrid grids are employed, which combine an unstructured grid representation in the vicinity of the nozzle with a structured grid representation in the wake region of the flow field. Performance benchmarks are drawn from large-scale runs performed using this solver, including a jet nozzle with chevrons and multi-species flows involving jet engine exhaust.

[1]Work sponsored by ONR

4:53PM L5.00007 Numerical study on the onset of electro-convection between planar electrodes caused by unipolar charge injection[1] , DOLFRED VIJAY FERNANDES, SURESH ALAPATI, YONG KWEON SUH, Dong-A University — Two-dimensional electrohydrodynamic (EHD) convection in a dielectric liquid between a pair of planar electrodes has been studied numerically considering autonomous unipolar charge injection from the bottom electrode. The steep gradient in the charge-density distribution and its time evolution can be resolved successfully using upwind schemes for the convective terms in the charge conservation equation. The Navier-Stokes equations coupled with the charge conservation equation and the Poisson equation for the potential are solved using the SIMPLE algorithm. We also apply a perturbation method to study the linear stability problem to derive more accurate critical parameter values for the onset of electro-convection. In particular, we are interested in the effect of injection strength and the horizontal length of the domain on the stability characteristics. Our two-dimensional numerical solutions are compared with the linear stability analysis for the validation of the code.

[1]This work was supported by NRF grant No. 2009-0083510 through Multi-phenomena CFD Engineering Research Center.

5:06PM L5.00008 Vorticity confinement technique for drag prediction , ALEX POVITSKY, TROY SNYDER, University of Akron — This work couples wake-integral drag prediction and vorticity confinement technique (VC) for the improved prediction of drag from CFD simulations. Induced drag computations of a thin wing are shown to be more accurate than the more widespread method of surface pressure integration when compared to theoretical lifting-line value. Furthermore, the VC method improves trailing vortex preservation and counteracts the shift from induced drag to numerical entropy drag with increasing distance of Trefftz plane downstream of the wing. Accurate induced drag prediction via the surface integration of pressure barring a sufficiently refined surface grid and increased computation time. Furthermore, the alternative wake-integral technique for drag prediction suffers from numerical dissipation. VC is shown to control the numerical dissipation with very modest computational overhead. The 2-D research code is used to test specific formulations of the VC body force terms and illustrate the computational efficiency of the method compared to a "brute force" reduction in spatial step size. For the 3-D wing simulation, ANSYS FLUENT is employed with the VC body force terms added to the solver with user-defined functions (UDFs). VC is successfully implemented to highly unsteady flows typical for Micro Air Vehicles (MAV) producing oscillative drag force either by natural vortex shedding at high angles of attack or by flapping wing motion.

5:19PM L5.00009 Computational and experimental study of airflow around a fan powered UVGI lamp , SRIKAR KALIGOTLA, Syracuse University, BEHTASH TAVAKOLI, Clarkson University, MARK GLAUSER, Syracuse University, GOODARZ AHMADI, Clarkson University — The quality of indoor air environment is very important for improving the health of occupants and reducing personal exposure to hazardous pollutants. An effective way of controlling air quality is by eliminating the airborne bacteria and viruses or by reducing their emissions. Ultraviolet Germicidal Irradiation (UVGI) lamps can effectively reduce these bio-contaminants in an indoor environment, but the efficiency of these systems depends on airflow in and around the device. UVGI lamps would not be as effective in stagnant environments as they would be when the moving air brings the bio-contaminant in their irradiation region. Introducing a fan into the UVGI system would augment the efficiency of the system's kill rate. Airflows in ventilated spaces are quite complex due to the vast range of length and velocity scales. The purpose of this research is to study these complex airflows using CFD techniques and validate computational model with airflow measurements around the device using Particle Image Velocimetry measurements. The experimental results including mean velocities, length scales and RMS values of fluctuating velocities are used in the CFD validation. Comparison of these data at different locations around the device with the CFD model predictions are performed and good agreement was observed.

5:32PM L5.00010 Integration Techniques for Stokes Boundary Integrals , MIKE NICHOLAS, Carthage College, TULANE UNIVERSITY CENTER FOR COMPUTATIONAL SCIENCE COLLABORATION — Stokes flow is often approached through a boundary integral formulation. The integrals involved are singular or nearly singular and therefore are not resolved well through standard quadrature methods. Various techniques to improve the accuracy of such integrals exist. We will present a brief survey of methods for singular and nearly singular Stokes integrals for both 2D and 3D flows.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L6 Drops VIII: Complex Phenomena 309

3:35PM L6.00001 Foliar disease transmission: insights from fluid dynamics , TRISTAN GILET, Universite de Liege, Belgium, LYDIA BOUROUIBA, JOHN BUSH, MIT — Rainfalls are suspected to trigger the spread of a multitude of foliar diseases that could be devastating for agricultural and forestry outputs and balance. A wealth of key fluid mechanics phenomena arise from the impact of drops on plant leaves. We present the results of a combined experimental and modelling investigation shading light on the modes of precipitation-induced foliar disease transmission.

3:48PM L6.00002 Sliding motion of an oscillating drop driven by ac electrowetting on inclined plane , JIWOO HONG, SEUNGJUN LEE, BONCHULL KOO, Department of Mechanical Engineering, POSTECH, YOUNG KWEON SUH, Department of Mechanical Engineering, Dong-A University, KWAN HYOUNG KANG, Department of Mechanical Engineering, POSTECH — When a drop is placed on an inclined solid plane, the drop can stick to or slide along the solid surface due to an opposition between the gravitational force and the pinning force from contact angle hysteresis. Here we demonstrate that sessile drops of sizes below the capillary length can be mobilized through drop oscillation induced by ac electrowetting at low frequency on a low inclined solid plane. The effects of ac frequency on the sliding condition and terminal sliding velocity are investigated. At resonant frequency, drops reach maximum terminal sliding velocities. Varying the applied voltage at a fixed frequency, we find the threshold voltage for a sliding drop and the empirical relationship between the applied voltage and terminal sliding velocity. Using the relationship between drop size and frequency, we can selectively slide drops of a specific size along an inclined plane.

4:01PM L6.00003 Droplet dynamics in microfluidic stratifications[1] , THOMAS CUBAUD, BIBIN M. JOSE, SAMIRA DARVISHI, RUOPENG SUN, Stony Brook University — We experimentally study the dynamic response of viscous droplets to external microflows. A segmented flow of droplets is injected into a stratified flow of miscible fluids using a square microchannel with two focusing sections. This method enables us to locally manipulate the capillary number of the continuous phase and examine the hydrodynamic coupling between droplets and a variety of flow fields in confined geometries. Focus is on the behavior of droplets entering a region with parallel streams having different interfacial tensions and viscosities. A general phase-diagram is proposed and droplet behavior is investigated in relation with dynamic wetting properties. This study shows methods for modifying the physicochemical environment of microfluidic droplets.

[1]This work is supported by NSF (CBET-0932925)

4:14PM L6.00004 Scaling laws for electrospraying , KRISHNARAJ SAMBATH, Purdue University, ROBERT COLLINS, Oak Ridge National Laboratory, MICHAEL HARRIS, OSMAN BASARAN, Purdue University — When stressed by strong electric fields, fluid menisci develop conical structures called Taylor cones which emit from their tips fine jets that in turn break up to form a mist of charged droplets. This phenomenon, known as electrohydrodynamic tip-streaming, cone-jetting, or electrospraying finds application in a number of industrially and scientifically important processes, including mass spectrometry, particle synthesis, and cloud physics. Here, this phenomenon is analyzed using the leaky-dielectric model developed by Taylor and Melcher for the situation in which an initially spherical free drop of an incompressible Newtonian fluid surrounded by a gas is subjected to an electric field. The resulting initial-boundary-value-problem is solved using the finite element method with elliptic mesh generation and adaptive, implicit time integration. The simulation results and simple scaling arguments are then used to infer the universal scaling laws governing the sizes and charges of the small electrospray drops.

4:27PM L6.00005 Transient Reduction of the Drag Coefficient of Charged Droplets via the Convective Reversal of Stagnant Caps , B.S. HAMLIN, W.D. RISTENPART, University of California at Davis — Droplets are frequently observed to move as if they were solid rather than liquid, i.e. with no slip at the liquid-liquid interface. This behavior is usually explained in terms of the so-called "stagnant cap" model, in which surfactants accumulate at the trailing edge of the droplet, immobilizing the surface and increasing the observed drag coefficient. Here we show that the drag coefficient for charged droplets is temporarily reduced by reversing the direction of an electric driving force. Using high speed video we simultaneously track the velocity and relative interfacial velocity of individual aqueous droplets moving electrophoretically through oil. The observed surface behavior is highly sensitive to the surfactant concentration. For sufficiently low or sufficiently high surfactant concentrations, upon reversal of the electric field the droplet rapidly accelerates in the opposite direction but then decelerates, concurrent with a transient rearrangement of tracer particles on the droplet surface. In contrast, droplets with intermediate surfactant concentrations exhibit no deceleration nor tracer particle rearrangement. We interpret these observations in terms of convectively dominated rearrangement of the stagnant cap, and we discuss the implications for precise electrophoretic control of droplet motion in lab-on-a-chip devices as well as droplet charge estimation through velocimetry.

4:40PM L6.00006 Electromagnetic liquid pistons for capillarity-based pumping , BERNARD MALOUIN, JOSEPH OLLES, LILI CHENG, AMIR HIRSA, Rensselaer Polytechnic Institute, MICHAEL VOGEL, Independent Researcher, Voorhees, NJ 08043 — Two adjoining ferrofluid droplets can behave as an electronically-controlled oscillator or switch by an appropriate balance of magnetic, capillary, and inertial forces. Their motion can be exploited to displace a surrounding liquid, forming electromagnetic liquid pistons. Such ferrofluid pistons can pump a precise volume of liquid via finely tunable amplitudes or resonant frequencies with no solid moving parts. Here we demonstrate the use of these liquid pistons in capillarity-dominated systems for variable focal distance liquid lenses with nearly perfect spherical interfaces. These liquid/liquid lenses feature many promising qualities not previously realized together in a liquid lens, including large apertures, immunity to evaporation, invariance to orientation relative to gravity, and low driving voltages. The dynamics of these liquid pistons is examined, with experimental measurements showing good agreement with a spherical cap model. A centimeter-scale lens was shown to respond in excess of 30 Hz, with resonant frequencies over 1 kHz predicted for scaled down systems.

4:53PM L6.00007 Negative radiation forces and the asymmetry of scattered radiation: spheres in Bessel beams , PHILIP L. MARSTON, LIKUN ZHANG, Washington State University — The discovery that acoustical [1] and optical [2,3] radiation forces computed on spheres placed on the axis of acoustical and optical Bessel beams may be opposite the direction of beam propagation makes it appropriate to reexamine the relationship between radiation forces and the asymmetry of the scattered radiation. For all of the previously identified acoustical cases in which the force was negative and the scattering pattern was also computed, it was found that the backscattering was suppressed and the forward scattering relatively enhanced (see e.g. [1]). In the present research the acoustic radiation force on an arbitrary isotropic sphere is related to the asymmetry in the scattering and the extinction introduced by the sphere for the case of a helical Bessel beam of arbitrary order [4]. The analysis confirms that conditions are more favorable for generating negative forces when the asymmetry is such that the backscattering is suppressed relative to the forward scattering. It is also found, however, that absorption of power by the sphere gives rise to a positive force contribution, a term which has been neglected in the corresponding optical analysis [2].
[1] P. L. Marston, J. Acoust. Soc. Am 120, 3518 (2006). [2] J. Chen et al., Nature Photonics (2011). [3] A. Novitsky & C.-W. Qiu, arXiv:1102.5285v1 (2011). [4] L. K. Zhang & P. L. Marston (submitted).

5:06PM L6.00008 Pulsating Electrohydrodynamic Cone-Jets: from Choked Jet to Oscillating Cone , DAVID BOBER, CHUAN-HUA CHEN, Duke University — Pulsating cone-jets occur in a variety of electrostatic spraying and printing systems. We report an experimental study of the pulsation frequency to reconcile two models based on a choked jet and an oscillating cone, respectively. The two regimes are demarcated by the ratio of the supplied flow rate (Q_s) to the minimum flow rate (Q_m) required for a steady Taylor cone-jet. When $Q_s < Q_m$, the electrohydrodynamic flow is choked at the nozzle because the intermittent jet, when on, emits mass at the minimum flow rate; the pulsation frequency in the choked jet regime is proportional to Q_s/Q_m. When $Q_s > Q_m$, the Taylor cone anchored at the nozzle experiences a capillary oscillation analogous to the Rayleigh mode of a free drop; the pulsation frequency in the oscillating cone regime plateaus to the capillary oscillation frequency which is independent of Q_s/Q_m.

5:19PM L6.00009 Electrohydrodynamic drop deformation by a strong electric field , DOV RHODES, EHUD YARIV, Technion — Using matched asymptotic expansions within the Taylor-Melcher leaky-dielectric model, we analyze the strong deformation of a drop by a strong electric field. As is common in slender-body analyses, the small drop slenderness is used as the expansion parameter. This parameter however is not a priori specified in the problem formulation, and must be found throughout the course of the asymptotic solution. Sherwood's scaling for dielectric liquids, inverse with the 6/7-power of the electric field, applied here. Slender shapes are possible only for low drop viscosities. We identify a new inequality, in terms of material-property ratios, necessary for the evolution to a slender shape, which is independent of Taylor discriminating inequality, necessary for initial deviation to prolate shapes.

5:32PM L6.00010 Desiccation of a Sessile Drop of Blood: Cracks Formation and Delamination , BENJAMIN SOBAC, IUSTI lab, Aix-Marseille Université, DAVID BRUTIN, IUSTI Lab, Aix-Marseille Univesité — The evaporation of drops of biological fluids has been studied since few years du to several applications in medical fields such as medical tests, drug screening, biostabilization... The evaporation of a drop of whole blood leads to the formation of final typical pattern of cracks. Flow motion, adhesion, gelation and fracturation all occur during the evaporation of this complex matter. During the drying, a sol-gel transition develops [1]. The drying kinetics is explained by a simple model of evaporation taking account of the evolution of the gelation front. The system solidifies and when stresses are too important, cracks nucleate. The cracks formation and the structure of the crack pattern are investigated. The initial crack spacing is found in good agreement with the implementation in open geometry of the model of cracks formation induced by evaporation proposed by Allain and Limat [2]. Finally, the drop is still drying after the end of the formation of cracks which leads, like in the situation of colloid suspensions [3], to the observation of a delamination phenomenon.
[1] B. Sobac and D. Brutin, Structural and Evaporative Evolutions in Desiccating Sessile Drops of Blood, Phys. Rev. E 84, 011603, 2011. [2] C. Allain and L. Limat, Phys. Rev. Lett. 74, 2981 (1995). [3] L. Pauchard, B. Abou, K. Sekimoto, Infuence of Mechanical Properties of Nanoparticles on Macrocrack Formation, Langmuir, 25(12), 6672-6677, 2009.

Monday, November 21, 2011 3:35PM - 5:45PM – Session L7 Turbulent Boundary Layers VI 310

3:35PM L7.00001 Three-dimensional structure of momentum transfer in turbulent channels[1] , ADRIÁN LOZANO-DURÁN, U. Politecnica de Madrid, ÓSCAR FLORES, U. Carlos III Madrid, JAVIER JIMÉNEZ, U. Politecnica de Madrid — The three-dimensional structures of the intense tangential Reynolds stress in plane turbulent channels (Qs) are studied by quadrant analysis, with emphasis on the logarithmic and outer layers. Wall-detached Qs are isotropically oriented background stress fluctuations, common to most turbulent flows, and do not contribute to the mean stress. Most of the stress is carried by a self-similar family of larger attached Qs, increasingly complex away from the wall, with fractal dimensions $D \approx 2$. They are "sponges of flakes," while vortex clusters are "sponges of strings." Although their number decays away from the wall, the fraction of the stress that they carry is independent of their heights, and a substantial part resides in a few objects extending beyond the centreline, reminiscent of the VLSM of several authors. The predominant logarithmic-layer structures are side-by-side Q4-Q2 pairs of sweeps and ejections, with an associated cluster, with dimensions and stresses similar to Townsend's conjectured attached eddies. They align themselves streamwise, but not strongly enough to explain the very long structures in the channel centre.

[1]Funded by CICYT, ERC and the Spanish Ministry of Science.

3:48PM L7.00002 Hairpin vortices in the transitional and developed turbulence of a flat-plate boundary layer[1] , GEORGE ILHWAN PARK, Stanford Univ., JAMES WALLACE, Univ. of Maryland, XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, Stanford Univ. — Using Vortex lines to reveal vortical structures in turbulent flows has been in disfavor for some time. They are field lines that can be drawn wherever the flow is rotational, regardless of whether a true vortex exists there or not. However, their virtues are that the vortices they can reveal do not depend on setting a detection threshold, unlike all the vortex identifiers based on the velocity gradient tensor or based on a low pressure criterion, and they can isolate individual vortices. Such individual hairpin vortices have been identified in the transition region near $Re_\theta = 500$ of a recent flat-plate boundary layer and in the developed turbulence near $Re_\theta = 1950$. The vortices in these two regions emerge out of sheets of unorganized vorticity in the viscous sublayer and have quite similar characteristics. An octant analysis based on the combinations of signs of the velocity and temperature fluctuations, u, v and θ, shows that the momentum and heat fluxes in both the transitional and developed regions are predominantly of the mean gradient type. The fluxes appear to be closely associated with wall layer vortices that transport momentum and heat toward and away from the wall.

[1]Supported by the CTR 2010 Summer Program.

4:01PM L7.00003 The structure of the vorticity field in the near-wall region of turbulent channel flow at high-Reynolds number , KOJI MORISHITA, TAKASHI ISHIHARA, Nagoya University, JST CREST, YUKIO KANEDA, Nagoya University — The structure of the vorticity field in turbulent channel flow is studied by using direct numerical simulations of the incompressible Navier-Stokes equations with up to $2048 \times 1536 \times 2048$ grid points; the maximum friction Reynolds number is $Re_\tau = 2560$. Instantaneous vortex-line plots show the presence of Ω-shaped hairpin vortices in the near-wall region of turbulent channel flow. The Ω-shaped hairpin vortices in the near-wall region are well displayed by a bundle of vortex lines starting from points along a line near $y^+ = 10$ parallel to the mean stream. They suggest that the hairpin vortices are formed by instabilities and roll-ups of sheets of spanwise vorticity in the buffer layer of turbulent channel flow. The three-dimensional structure of the low-speed region of the streamwise velocity near the wall ($y^+ < 100$) is discussed in view of these vortex lines starting from the buffer layer.

4:14PM L7.00004 Flow Visualization of Artificially Generated Hairpin Vortices , DANIEL SABATINO, Lafayette College, MARK PALFRAMAN, Virginia Tech — To investigate the potential mechanisms for hairpin packet formation in fully turbulent boundary layers, a flow visualization study of artificially generated hairpin vortices in an otherwise laminar boundary layer is performed. The experiments are conducted in a recently constructed free surface water channel at Lafayette College. A new method to artificially generate individual hairpin vortices is employed which utilizes a flexible membrane which is inflated to create transient hemispherical protrusions on a flat plate, zero pressure gradient laminar boundary layer. By controlling the duration of time the membrane protrudes above the wall, a single vortex can be reliably generated. This technique avoids the need for fluid injection in order to ensure uniform particle seeding for subsequent PIV measurements. Multiple generation sites are placed at different streamwise locations to allow hairpins of different maturity to interact. The characteristics of single hairpin vortices will be compared to those described in the literature along with a qualitative analysis of the interaction of two hairpin vortices.

4:27PM L7.00005 Dynamical Properties of Vortex Furrows in Transitioning Boundary Layers , PETER BERNARD, University of Maryland — A vortex filament simulation of the spatially growing transitional boundary layer reveals the presence of low speed streaks underlying furrow-like streamwise oriented folds in the surface vorticity layer (AIAA J. Vol. 48, 2010; Proc. ETC13, 2011). The putative hairpin vortices and packets widely observed in boundary layers are found to be an illusion created by assigning the status of structure to the visualized form of regions of rotational motion created by the vortex furrows. Thus, at best, hairpins roughly describe the shape taken by that part of the vorticity within the furrows that directly causes rotation while ignoring the "invisible" and considerable non-rotational part. The life history of the furrows is discussed here including a description of how they grow and the dynamics of the vorticity field within them. Long lived furrows represent "factories" within which initially spanwise vorticity progresses from arch to either one or two-lobed mushroom-like structures in a continuous stream. Furrows grow by this same process. At the heart of the furrow phenomenon is a self-reinforcing process by which streamwise vorticity begets more streamwise vorticity.

4:40PM L7.00006 Evolution of Reynolds stresses in a turbulent boundary layer , ROELAND DE KAT, University of Southampton, LIAN GAN, JAMES DAWSON, University of Cambridge, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — Understanding Reynolds shear-stress events in a turbulent boundary layer is of crucial importance for modelling and controlling turbulent wall-flows. In this study, we examine the evolution in time and space of these shear-stress events by performing time-resolved PIV measurements in a stream-wise wall-normal plane of a turbulent boundary layer at $Re_\tau \approx 2500$. The conditions are similar to the experiment by Dennis & Nickels (J. Fluid Mech. 2011, vol. 673), who performed measurements at $Re_\theta = 4700$. Four high-speed cameras positioned next to each other, 4-5 m downstream of a glass rod trip, captured a region of flow spanning approximately 2δ in stream-wise and 0.5δ in wall-normal direction. This zoomed-in field-of-view enables high spatial, $l^+ \approx 20$, and temporal resolution, $\Delta t^+ \approx 1$ which will allow us to describe the evolution of shear-stress events in time and space. In the talk, detailed analyses including instantaneous tracking of Reynolds shear-stress events, quadrant decomposition and spectra of the stream-wise, wall-normal and Reynolds shear-stress fluctuations will be presented.

4:53PM L7.00007 The topology of the footprints of wall-turbulence , JASON MONTY, University of Melbourne, MIN CHONG, IVAN MARUSIC, The University of Melbourne — When studying the topology of turbulent flows, the three invariants of the velocity gradient tensor are often used. For incompressible flow the first invariant P is zero and the topology of the flow structures can be investigated in terms of the second and third invariants, Q and R respectively. For example, isosurfaces of Q above a certain threshold are often used in an attempt to identify vortical structures in the flow. In wall-turbulence, however, these invariants are zero on a no slip wall. Therefore, analysis tools relying on these invariants cannot be used to topologically study the footprint of turbulence on the wall. In this paper, it is proposed that the "flow" field on a wall can be described by a no slip Taylor-series expansion. This provides a new tensor relating skin friction to streamwise and spanwise coordinate. Like the velocity gradient tensor, it is possible to define invariants $\mathcal{P}$, $\mathcal{Q}$ and $\mathcal{R}$ of the so-called "no slip" tensor. It will also be shown that it may be possible to investigate the topology of the flow field on a no slip wall in terms of these invariants.

5:06PM L7.00008 Space-Time Correlation of Large-Scale Structures in a Turbulent Boundary Layer[1] , NICOLAS BUCHMANN, CALLUM ATKINSON, MATTHIAS KUEHN, JULIO SORIA, Department of Mechanical and Aerospace Engineering, Monash University, Victoria 3800, Australia — Taylor's hypothesis is often used to project temporal data into the spatial domain and has been used in the past to show the presence of large-scale structures ($> 10\delta$) in the log and lower wake region of the turbulent boundary layer (TBL). To investigate the spatial and temporal evolution of such large-scale structures, the present study employs time-resolved Particle Image Velocimetry (PIV) in several streamwise-spanwise planes in the log-layer of a TBL ($Re_\theta = 2,000$). In order to capture the full extent of these structures, four high-speed, high-resolution PIV systems are combined to span a region of approximately $3\delta \times 12\delta$ and a continuous time sequences of $\approx 50\delta/U$. Such data sets are currently unavailable from previous experimental investigations and reveal the existence of long and very long ($> 8\delta$) low- and high-speed structures. Two-point space-time correlations are employed to examine the temporal extent and meandering nature of these structures with respect to their size and spacing in the log-layer. Furthermore, the validity of Taylor's hypothesis is tested for such long projection distances.

[1]This work is supported by the Australian Research Council through a Discovery and LIEF grant.

5:19PM L7.00009 High spatial resolution measurements of large-scale three-dimensional structures in a turbulent boundary layer[1] , CALLUM ATKINSON, NICOLAS BUCHMANN, MATTHIAS KUEHN, JULIO SORIA, Monash University — Large-scale three-dimensional (3D) structures in a turbulent boundary layer at $\mathrm{Re}_\theta = 2000$ are examined via the streamwise extrapolation of time-resolved stereo particle image velocimetry (SPIV) measurements in a wall-normal spanwise plane using Taylor's hypothesis. Two overlapping SPIV systems are used to provide a field of view similar to that of direct numerical simulations (DNS) on the order of $50\delta \times 1.5\delta \times 3.0\delta$ in the streamwise, wall-normal and spanwise directions, respectively, with an interrogation window size of $40^+ \times 20^+ \times 60^+$ wall units. Velocity power spectra are compared with DNS to examine the effective resolution of these measurements and two-point correlations are performed to investigate the integral length scales associated with coherent velocity and vorticity fluctuations. Individual coherent structures are detected to provide statistics on the 3D size, spacing, and angular orientation of large-scale structures, as well as their contribution to the total turbulent kinetic energy and Reynolds shear stress.

[1]The support of the ARC through Discovery (and LIEF) grants is gratefully acknowledged

5:32PM L7.00010 Energy dissipating structures in turbulent boundary layers , MARIE FARGE, Ecole Normale Superieure, Paris, ROMAIN NGUYEN VAN YEN, Freie Universitaet, Berlin, KAI SCHNEIDER, Aix-Marseille Universite — We present numerical experiments of a dipole crashing into a wall, a generic event in two-dimensional incompressible flows with solid boundaries. The Reynolds number Re is varied from 985 to 7880, and no-slip boundary conditions are approximated by Navier boundary conditions with a slip length proportional to Re^{-1}. Energy dissipation is shown to first set up within a vorticity sheet of thickness proportional to Re^{-1} in the neighborhood of the wall, and to continue as this sheet rolls up into a spiral and detaches from the wall. The energy dissipation rate integrated over these regions appears to converge towards Rey-independent values, indicating the existence of energy dissipating structures that persist in the vanishing viscosity limit. Details can be found in Nguyen van yen, Farge and Schneider, PRL, **106**, 184502 (2011).

Monday, November 21, 2011 3:35PM - 5:45PM – Session L8 Compressible Flows II 311

3:35PM L8.00001 Low-dissipation hybrid schemes for simulations of compressible multicomponent flows , POOYA MOVAHED, ERIC JOHNSEN, University of Michigan, Ann Arbor — In the present work an efficient hybrid scheme is proposed for numerical simulations of compressible multicomponent flows. The algorithm is based on a high-order accurate weighted essentially non-oscillatory (WENO) scheme for shock capturing and a non-dissipative central scheme in the split form for smooth regions. The central-difference method results in a reasonable speed up and exhibits better resolution properties for turbulence. The shock capturing is handled using the AUSM+up Riemann solver with a WENO reconstruction of the primitive variables. A new sensor based on the first norm of the difference of WENO weights from the ideal weights is used at the beginning of each Runge-Kutta step for a smooth transition between the central and WENO fluxes at interfaces. The scheme is shown to prevent spurious pressure oscillations at interfaces. The performance of the method is presented for a set of problems including the Sod shock tube problem, the Shu-Osher problem and the planar Richtmyer-Meshkov instability with particular emphasis on mixing at early and late times. This research was supported in part by the DOE NNSA under the Predictive Science Academic Alliance Program by grant DEFC52- 08NA28616.

3:48PM L8.00002 A Novel A Posteriori Investigation of Scalar Flux Models for Passive Scalar Dispersion in Compressible Boundary Layer Flows , KALEN BRAMAN, VENKAT RAMAN, The University of Texas at Austin — A novel direct numerical simulation (DNS) based a posteriori technique has been developed to investigate scalar transport modeling error. The methodology is used to test Reynolds-averaged Navier-Stokes turbulent scalar flux models for compressible boundary layer flows. Time-averaged DNS velocity and turbulence fields provide the information necessary to evolve the time-averaged scalar transport equation without requiring the use of turbulence modeling. With this technique, passive dispersion of a scalar from a boundary layer surface in a supersonic flow is studied with scalar flux modeling error isolated from any flowfield modeling errors. Several different scalar flux models are used. It is seen that the simple gradient diffusion model overpredicts scalar dispersion, while anisotropic scalar flux models underpredict dispersion. Further, the use of more complex models does not necessarily guarantee an increase in predictive accuracy, indicating that key physics is missing from existing models. Using comparisons of both a priori and a posteriori scalar flux evaluations with DNS data, the main modeling shortcomings are identified. Results will be presented for different boundary layer conditions.

4:01PM L8.00003 The conservative cascade of kinetic energy in compressible turbulence , HUSSEIN ALUIE, SHENGTAI LI, HUI LI, Los Alamos National Laboratory — We use a coarse-graining approach to analyze inter-scale transfer of kinetic energy in compressible turbulence. We present the first direct evidence that mean kinetic energy cascades conservatively beyond a transitional "conversion" scale-range despite not being an invariant of the compressible flow dynamics. We use high-resolution three-dimensional simulations of compressible hydrodynamic turbulence on 512^3 and 1024^3 grids. We probe regimes of forced steady-state isothermal flows and of unforced decaying ideal gas flows. The key quantity we measure is pressure dilatation cospectrum, $E^{PD}(k)$, where we provide the first numerical evidence that it decays at a rate faster than k^{-1} as a function of wavenumber. This is sufficient to imply that mean pressure dilatation acts primarily at large-scales and that kinetic and internal energy budgets statistically decouple beyond a transitional scale-range. Our analysis establishes the existence of an ensuing inertial range over which mean SGS kinetic energy flux becomes constant, independent of scale. Over this inertial range, mean kinetic energy cascades locally and in a conservative fashion despite not being an invariant.

4:14PM L8.00004 Energy-Pressure-Velocity-Scalar Filtered Mass Density Function , MEHDI B. NIK, PEYMAN GIVI, University of Pittsburgh, CYRUS MADNIA, University at Buffalo (SUNY), STEPHEN B. POPE, Cornell University — The "Energy-Pressure-Velocity-Scalar Filtered Mass Density Function" (EPVS-FMDF) is a new subgrid scale (SGS) model developed for large eddy simulation of high speed turbulent flows. This is an extension of the previously developed "velocity-scalar filtered mass density function" method [1] in low speed flows. In the EPVS-FMDF formulation, compressibility effects are accounted for by including two additional thermodynamic variables: the pressure and the internal energy. This is the most general form of the FDF for high speed flow simulations. The EPVS-FMDF is obtained by solving its transport equation, in which the effects of convection for velocity and scalar field appear in a closed form. The unclosed terms are modeled in a fashion similar to that in RANS-PDF methods. The modeled EPVS-FMDF transport equation is solved by a Lagrangian Monte Carlo method and is employed for LES of a temporally developing mixing layer at several values of the convective Mach number. The predicted results are assessed by comparison with direct numerical simulation (DNS) data.

[1] Sheikhi, M. R. H., Givi, P., and Pope, S. B., Velocity-Scalar Filtered Mass Density Function for Large Eddy Simulation of Turbulent Reacting Flows, Phys. Fluids, 19(9): 095196 1-21 (2007)

4:27PM L8.00005 Small-scale intermittency in compressible turbulence , DIEGO DONZIS, SHRIRAM JAGANNATHAN, Texas A&M University — While the effects of compressibility on low-order quantities such as the mean turbulent kinetic energy and dissipation in decaying turbulence have been extensively investigated, little is known about the scaling of fine intermittent structures and how they scale with Reynolds and turbulent Mach numbers. It is thus unclear how the intermittent behavior of energy dissipation rate, for example, in compressible flows compares with its incompressible counterpart. Massive direct numerical simulations of isotropic compressible turbulence at finer-than-usual grid resolutions were conducted to investigate the scaling of small-scale intermittency at a range of Reynolds and turbulent Mach numbers. Large-scale forcing is applied to attain a stationary state which permits better statistical sampling of intermittent activity and, at the same time, provides results independent of initial conditions. Scaling exponents for energy dissipation rate are computed and compared to theoretical models. While low-order moments of dissipation show weak dependence on compressibility levels and thus possess scaling exponents similar to the incompressible case, high-order moments depend on the turbulent Mach number. Differences in the scaling of solenoidal and dilatational components are related to the structure of the most intense events for each type of motion. The consequences of the findings are discussed in the context of high Reynolds number flows.

4:40PM L8.00006 A numerical method for DNS/LES of high–enthalpy turbulent flows , SHANKAR GHOSH, KRISHNAN MAHESH, University of Minnesota — A numerical method is developed for simulation of high– enthalpy turbulent flows. A non-dissipative algorithm is used for accurate flux reconstruction at the cell faces. The method is combined with a predictor corrector based shock capturing scheme to simulate strong shock waves encountered in high– enthalpy flows. A non–linear limiter is used to limit the application of shock capturing only to the vicinity of the shock wave to minimize dissipation. The Navier-Stokes equations are suitably modified to represent various thermo–chemical processes occurring in high–enthalpy flows. A five species model for air is considered. To account for finite rate chemical reactions, individual mass conservation equations are solved for every species. An equation for conservation of vibrational energy is also solved to account for vibrational excitation. Species diffusion is modeled through Fick's law. Transport properties are computed taking high temperature effects into account. The numerical method is evaluated using test problems.

4:53PM L8.00007 Modeling Compressed Turbulence with BHR , DANIEL ISRAEL, Los Alamos National Laboratory — Turbulence undergoing compression or expansion occurs in systems ranging from internal combustion engines to supernovae. One common feature in many of these systems is the presence of multiple reacting species. Direct numerical simulation data is available for the single-fluid, low turbulent Mach number case. Wu, et al. (1985) compared their DNS results to several Reynolds-averaged Navier-Stokes models. They also proposed a three-equation $k-\varepsilon-\tau$ model, in conjunction with a Reynolds-stress model. Subsequent researchers have proposed alternative corrections to the standard $k-\varepsilon$ formulation. Here we investigate three variants of the BHR model (Besnard, 1992). BHR is a model for multi-species variable-density turbulence. The three variants are the linear eddy-viscosity, algebraic-stress, and full Reynolds-stress formulations. We then examine the predictions of the model for the fluctuating density field for the case of variable-density turbulence.

5:06PM L8.00008 Differential Reynolds stress closure modeling of compressible shear flows , CARLOS GOMEZ, SHARATH GIRIMAJI, Texas A&M University — The most important difference between turbulence in high and low Mach number flows stems from the changing role and action of pressure at different speed regimes. In the rapidly distorted turbulence regime, this effect is well captured by rapid distortion theory (RDT) which shows that gradient Mach number characterizes the role/action of pressure very accurately. Thus motivated, we develop a new rapid pressure-strain correlation model for high-speed compressible shear flows in which the closure coefficients are functions of the local gradient Mach number. The functional dependence of the model coefficients on the Mach number is obtained by comparison against RDT data. This closure naturally leads to a pressure-dilatation closure model. Further analysis reveals that a modification of the dissipation equation is also mandatory to accommodate the pressure-dilatation closure physics. Full differential Reynolds stress closure calculations of plane supersonic mixing layers are performed and comparison with the experimental data of Goebel and Dutton shows that the model exhibits good overall agreement without any further model calibration.

5:19PM L8.00009 *A Priori* Assessment of the FDF Sub-Closures in Compressible Turbulence , NAVID S. VAGHEFI, SUNY at Buffalo, MEHDI B. NIK, PATRICK PISCIUNERI, PEYMAN GIVI, University of Pittsburgh, CYRUS K. MADNIA, SUNY at Buffalo — Results are presented of *a priori* assessment of some of the sub-closures required in LES via FDF in compressible turbulent flows. This is done via assessment of DNS of several turbulent flow configurations at varying compressibility levels and Reynolds numbers. Optimum model parameters are calculated by maximizing the correlation coefficients between the SGS exact and modelled terms. The effects of the filter width are also assessed for some of the sub-closures.

5:32PM L8.00010 Application of gradient limiters for computation of viscous fluxes in an unstructured compressible flow solver , J.P. STRODTBECK, Unversity of Kentucky, K. WEBER, Rolls-Royce LLC, J.M. MCDONOUGH, University of Kentucky — HYDRA, an unstructured finite-volume CFD code used internally by Rolls–Royce LLC, evaluates viscous fluxes using a characteristic-based scheme in which the characteristic variables are modified with a pseudo-Laplacian smoothing introduced in the doctoral dissertation of Moinier (Oxford University, 1999). Since the pseudo-Laplacian scheme is inadequate for removing numerical oscillations in a variety of situations, a replacement scheme is proposed and implemented with characteristic variables approximated using a smoothed flux limiter based on a traditional minmod scheme. Formally, the method retains second-order accuracy except near oscillations. Convergence plots and comparisons with data demonstrate that the limiter technique provides improvement compared with baseline simulations. Convergence plot comparisons show improved mass flow conservation, removal of oscillations, and the capability of converging to machine zero without sacrificing overall accuracy. Besides this specific application to shock capturing in compressible flows, similar flux limiters may also be appropriate for use in implicit LES for incompressible flows where other limiters and/or filters are currently used in a similar pseudo-Laplacian manner, and also for compressible LES.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L9 Flow Control IV 312

3:35PM L9.00001 Unsteady Aerodynamic Flow Control of a Suspended Axisymmetric Moving Platform[1] , THOMAS LAMBERT, BOJAN VUKASINOVIC, ARI GLEZER, Georgia Tech — The aerodynamic forces on an axisymmetric wind tunnel model are altered by fluidic interaction of an azimuthal array of integrated synthetic jet actuators with the cross flow. Four-quadrant actuators are integrated into a Coanda surface on the aft section of the body, and the jets emanate from narrow, azimuthally segmented slots equally distributed around the model's perimeter. The model is suspended in the tunnel using eight wires each comprising miniature in-line force sensors and shape-memory-alloy (SMA) strands that are used to control the instantaneous forces and moments on the model and its orientation. The interaction of the actuation jets with the flow over the moving model is investigated using PIV and time-resolved force measurements to assess the transitory aerodynamic loading effected by coupling between the induced motion of the aerodynamic surface and the fluid dynamics that is driven by the actuation. It is shown that these interactions can lead to effective control of the aerodynamic forces and moments, and thereby of the model's motion.

[1]Supported by ARO

3:48PM L9.00002 Experimental Study of Plasma Control of an Unstarting Supersonic Flow[1] , SEONG-KYUN IM, Stanford University, HYUNGROK DO, University of Notre Dame, MARK A. CAPPELLI, Stanford University — Experimental studies of the control of unstarting supersonic model inlet flows using Dielectric Barrier Discharges (DBD) is demonstrated at Mach 4.7 flow conditions and a static temperature of $\sim$60K and static pressure of $\sim$1kPa. Planar Laser Rayleigh Scattering (PLRS) is used to visualize important flow features, such as boundary layers and shockwaves. Supersonic flow unstart is initiated by injecting mass into model inlet flows of either laminar or tripped turbulent boundary layer flow conditions. DBD discharge actuation of the tripped turbulent flow delays the unstart process, shifting the unstart dynamics closer to what is seen for the laminar boundary layer case. In all studies, a single DBD actuator pair is used, oriented parallel to the freestream flow, generating spanwise disturbances. It is proposed that strong suction flow which brings high momentum freestream flow near exposed electrode can be a mechanism of this actuation. PLRS reveals that this actuation is spatially confined to the regions close to the actuator electrodes, greatly limiting their performance.

[1]This material is based upon work supported by the Department of Energy under Award Number(s)DE-FC52-08NA28614.

4:01PM L9.00003 High Mach Number Leading-edge Flow Separation Control using AC DBD Plasma Actuators[1] , CHRISTOPHER KELLEY, PATRICK BOWLES, JOHN COONEY, CHUAN HE, THOMAS CORKE, University of Notre Dame, BRADLEY OSBORNE, JOSEPH SILKEY, JOSEPH ZEHNLE, The Boeing Company — Wind tunnel experiments were conducted to quantify the effectiveness of alternating current dielectric barrier discharge flow control actuators to suppress leading-edge stall on a NASA energy efficient transport airfoil at compressible freestream speeds. The objective of this research was to increase lift, reduce drag, and improve the stall characteristics of the supercritical airfoil near stall by flow reattachment at relatively high Mach and Reynolds numbers. In addition, the effect of unsteady (or duty cycle) operation on these aerodynamic quantities was also investigated. The experiments were conducted for a range of Mach numbers between 0.1 and 0.4. corresponding to a Reynolds number range of 560,000 through 2,260,000. Lift, drag, quarter chord moment, and suction side pressures were measured near stall for baseline, steady actuation, and a scan of nondimensional duty cycle frequencies. The results show that the plasma actuators were effective at reattaching the leading-edge separated flow as evidenced by the increase in maximum lift coefficient and stall angle (as much as 2.5 degrees). The experiment also showed that lift was increased the most when the plasma actuator was operated unsteady with a nondimensional frequency of unity.

[1]The Boeing Company

4:14PM L9.00004 Interaction of Finite-span Synthetic-jets with a Cross-flow over a Swept Wing , JOSEPH VASILE, MICHAEL AMITAY, Rensselaer Polytechnic Institute — The formation of secondary flow structures due to the interaction of three finite span synthetic jets with a cross-flow was investigated experimentally over a finite sweptback wing (cross-sectional profile of the NACA 4421) at a Reynolds number of 100,000. Stereoscopic PIV data were collected across the three jets in the wing's mid-span section, where the effect of the jets' location, and their blowing ratio were analyzed based on the three-dimensional flow field using time-averaged and phase-averaged statistics. The arrangement of synthetic jets was investigated through the use of varying actuation combinations in order to fully understand the interaction of the three jets with the cross flow. In the present study, an angle of attack of 13.5deg was chosen for the model, in which the boundary layer was attached in the vicinity of the middle synthetic jet and partially separated in the vicinity of the jet closer to the wing tip. The present work confirmed the previous findings of the presence of secondary tilted flow structures.

4:27PM L9.00005 Reduction of vortex shedding intensity from a cylinder using semi-active flow control , JARLE V. EKANGER, Flow Design Bureau AS, MORTEN KJELDSEN, NTNU — Experiments were performed in the open water channel at the Waterpower Laboratory, NTNU, Norway, with the aim of reducing vortex shedding intensity by semi-active flow control. The test rig consisted of a perforated steel tube lined by a rubber bellows. The holes (d/D=0.6) formed a line at the leading edge, one tube diameter apart. Two flow control modes were attainable; (1) lining being flush with the cylinder wall, and (2) pressurized lining creating leading edge bumps. Upstream flow conditions were monitored, and used as input for the control loop governing the pressure of the lining. A flat metal rod, onto which strain gauges were glued, was positioned in the wake. It was assumed that the motion of the rod corresponded to the velocity components normal to the main flow direction. Thus the motion of the rod described the vortex shedding from the tube. Strouhal numbers were found to be approximately 0.3. It was the assumption that the bumps would disrupt vortex formation and reduce the vortex intensity. Tests showed that the assumption was plausible, with observed intensity reductions of 15-30% for $\mathrm{Re}_D \sim$ [20000 to 50000]. Plots also appear to show a breakdown of organization in the wake when the tube is in activated mode. It was shown that semi-active control of vortex shedding behind a cylinder is achievable.

4:40PM L9.00006 Lift and Drag on a NACA0015 Airfoil With Duty Cycle Active Flow Control[1] , POOYA KABIRI, DOUGLAS BOHL, GOODARZ AHMADI, Clarkson University — Active flow control experiments were carried out over a NACA 0015 airfoil with a trailing edge flap. Two arrays of synthetic jet actuators were mounted in the airfoil with one on near the leading edge (0.1c) and the other on the main wing body near the wing/flap interface (0.65c). Characterization of the SJA's showed they produced their highest exit velocities at a frequency of 1100 Hz, which was near the natural frequency of the piezo membranes. When actuated at frequencies corresponding to the flow natural frequencies (10-100Hz) the jets produced no jet velocity. In order to control the flow using a frequency near the flow's natural shedding frequency the synthetic jets were actuated using a forcing frequency near the piezo natural frequency with a duty cycle frequency of 10-1000Hz. Force balance results showed that for a 0° flap deflection the active flow control delayed stall and lowered drag regardless of the duty cycle frequency. At flap deflections of 20° and 40° differences were observed between the continuously forced and duty cycles cases. For these cases continuous forcing increased the stall angle and reduced drag. Duty cycle forcing also delayed stall however it significantly increased drag near the stall AOA even compared to the no forcing case.

[1]Research Supported by AFOSR Grant FA9550-09-1-0051

4:53PM L9.00007 Manipulating forces by interactive separation and circulation control , HASSAN NAGIB, PAUL REINHARD, PAUL ROZIER, IIT, Chicago — Steady, unsteady and intermittent suction and blowing from localized or distributed slots are used to reveal the physical mechanisms and their interaction in order to manipulate (enhance or reduce) the forces on various aerodynamic bodies and surfaces. Performance under ideal inviscid conditions is used as a standard of performance to compare the outcomes to. While high-lift airfoils were part of the focus, flow over humps which lead to large separation zones was also investigated. Surface pressure measurements, wake surveys and surface visualization were utilized over a wide range of operating conditions in the NDF at IIT. Velocities ranged from 20 to 110 m/s ($0.06 < M < 0.31$), corresponding to chord Reynolds numbers from $500,000$ to $3,700,000$, and included a full range of airfoil angles of attack with flap deflections from 10 to 55 degrees and various leading edge configurations. Steady suction control was more effective at eliminating the large separation bubble created by the model, requiring a pressure ratio between the applied force and inviscid force of approximately unity, whereas blowing required a two to one ratio. Pulsed suction was superior and enhanced by the operating frequency or duty cycle. Separation control (SC) was modified by the presence of circulation control (CC). Steady-blowing SC near the leading edge reduced the effect of blown-flap CC, whereas steady-suction SC increased the performance gain.

5:06PM L9.00008 Improvement for efficiency of frictional drag reduction by repetitive bubble injection , YUJI TASAKA, YUICHI MURAI, HYUN-JIN PARK, YUKIHIRO TOHGE, Hokkaido University — Repetitive bubble injection was examined for promoting frictional drag reduction by bubbles based on idea of "resource allocation" and voidage waves formed in a horizontal bubbly channel. Gain factor, which is defined as the ratio of degree of the drag reduction by the mean void fraction of bubbles, α, showed dramatic improvement on the efficiency of the drag reduction in comparison with continuous bubble injection at the same α. This improvement is prominent at small void fractions at which the continuous bubble injections enhance the frictional drag. Simultaneous recording of advecting bubbles with measurements of shear stress profiles at the channel wall indicated that preceding large bubbles provide great drag reduction and following small, low concentration bubbles don't affect the frictional drag. Namely the bubbles as a limited resource are efficiently used for the drag reduction in the repetitive bubble injection.

5:19PM L9.00009 Transitory Aerodynamic Control of Unsteady Separation[1] , GEORGE T.K. WOO, ARI GLEZER, Georgia Institute of Technology — The dynamics of the controlled three-dimensional attachment of transitory stall over an airfoil oscillating in pitch is investigated in wind tunnel experiments using a partial spanwise 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. It is shown that the control authority of the single pulse is highly-dependent on the oscillation cycle phase due to the timed interactions between the actuation jets and the evolution of the dynamic stall vortex on the suction surface. The transitory effects of the actuation can be extended and exploited for significant suppression of the dynamic stall by using successive pulsed actuation strategically staged during the cycle. High-resolution phase-locked PIV measurements in the cross stream plane on the suction surface and in the near wake demonstrate that the pulsed actuation sequence can effectively regulate the accumulation (trapping) and shedding of vorticity. The time-modulation of the vorticity fluxes results in significant temporal changes in circulation, and consequently in the measured time-dependent aerodynamic forces and moments.

[1]Supported by AFOSR.

5:32PM L9.00010 Experimental Study of the Temporal Nature of an Actively Controlled Three Dimensional Turret Wake , PATRICK SHEA, MARK GLAUSER, Syracuse University — Experimental measurements have been performed to characterize the actively controlled wake of a three-dimensional, non- conformal turret which is a bluff body commonly used for housing optical systems on airborne platforms. As a bluff body, turrets can generate strong turbulent flow fields that degrade the performance of the optical systems and the aircraft. Experiments were performed in a low-speed wind tunnel at Syracuse University using particle image velocimetry and dynamic pressure measurements with the objective of developing a better understanding of the spatial and temporal nature of the wake flow field. Active control was achieved using dynamic suction in the vicinity of the turret aperture and was found to have a significant impact on the structure of the wake as well as the temporal characteristics of the flow field. With a better understanding of the wake characteristics, closed-loop, active flow control systems will be developed to help reduce fluctuating loading and aero- optical distortions associated with the turbulent flow field.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L10 General Fluid Dynamics I 313

3:35PM L10.00001 ABSTRACT WITHDRAWN –

3:48PM L10.00002 The origin of the self-organization of the 2D Euler fluid flows[1] , FLORIN SPINEANU, MADALINA VLAD, Natl. Inst. Laser, Plasma Rad. Physics — The 2D ideal incompressible fluid is usually described in terms of streamfunction, velocity and vorticity. An equivalent model consists of a discrete set of point-like vortices interacting in plane by a long-range potential. The essential property of the latter model is that it re-formulates the description in terms of matter, field and interaction. We first extend the model to reflect the parity-invariance and show that returning to continuum it leads to a field-theoretical formulation, with a Lagrangian density for a nonlinear scalar (matter) field, a gauge field and their minimal interaction. A fundamental property of the 2D Euler fluid is revealed in this way: the extremum of the action functional shows Self-Duality, a property known to generate coherent structures (almost all known solitons and instantons in the natural systems). We derive analytically the sinh-Poisson equation, governing the stationary states at relaxation.The presence of the Chern- Simons part in the Lagrangian explains why in 3D the fluid will never relax to a stationary coherent flow. Connections with 4D fermion systems (Nambu-Jona-Lasinio) and with surfaces of constant mean curvature (CMC) will be presented. Stability of certain regular flows results from the property of non-self- intersection of CMC surfaces embedded in 3D space.

[1]Partially supported by CNCSIS-UEFISCDI, PNII - IDEI 1104/2008.

4:01PM L10.00003 On Liouville's theorem in fluid mechanics , P.J. MORRISON, The University of Texas at Austin, F. BOUCHET, (CNRS) ENS-Lyon, France, S. THALABARD, ENS-Lyon, France, O.V. ZABORONSKI, University of Warwick, Coventry, UK — Since the early work of Burgers it has been known that discretizations of fluid models possess a version of Liouville's theorem on conservation of phase space volume. In fact, spectral representations of two-dimensional turbulence are known to have a detailed version of this theorem. The existence of such Liouville theorems led many (e.g. Burgers, Lee, Kraichnan and Montgomery) to consider various statistical mechanical approaches to turbulence. We show how this theorem arises naturally from the Hamiltonian structure of inviscid fluid equations.

4:14PM L10.00004 A "reciprocal" theorem for the prediction of loads on a body moving in an inhomogeneous flow at arbitrary Reynolds number , JACQUES MAGNAUDET — We derive a theorem paralleling Lorentz's reciprocal theorem and providing general expressions for the force and torque acting on a rigid body of arbitrary shape moving in an inhomogeneous incompressible flow at arbitrary Reynolds number. This theorem follows the approach initiated by Quartapelle and Napolitano (1983) by making use of auxiliary solenoidal irrotational velocity fields. It allows any component of the force and torque to be evaluated solely in terms of velocity and vorticity, irrespective of its orientation with respect to the relative velocity between the body and fluid. When the body moves in a time-dependent linear flow, this theorem reveals the various couplings between the body translation and rotation and the strain rate and vorticity of the carrying flow. We show that the predictions obtained with this approach encompass all those previously obtained in the inviscid limit. We also show how it can be used to evaluate explicitly the drag and lift components of the force acting on high-Reynolds-number bubbles moving in inhomogeneous flows.

4:27PM L10.00005 Spatiotemporal cascades of the Poiseuille-Hagen flow in invariant elliptic structures , VICTOR MIROSHNIKOV, College of Mount Saint Vincent — Spatiotemporal cascades of the transitional Poiseuille-Hagen flow are considered in elliptic structures, which are invariant with respect to differential and nonlinear algebraic operations. Differentiation and algebra of the invariant structures and decomposition of smooth velocity profiles in the invariant structures are treated both theoretically and symbolically. Reduction of the invariant elliptic structures to invariant trigonometric structures and invariant hyperbolic structures is also considered. By using the invariant structures, the displayed and hidden perturbations of the basic Poiseuille-Hagen flow are represented as dual perturbations, while the series solution for the perturbed flow converges uniformly. The cascade solution for the Poiseuille-Hagen flow is constructed in a multiscale form, which explicitly shows the effect of various factors at multiple scales.

4:40PM L10.00006 An Exact Solution of the 3D Navier-Stokes Equation , AMADOR MURIEL, Harvard University — We use a time evolution equation for the single particle distribution function (Muriel, Physica D, 1997) to generate the time evolution of the velocity fields. These velocity fields are then substituted into the Navier-Stokes equation to calculate the pressure gradients. The pressure gradients are then integrated over space to generate pressure tensors. In addition, we calculate the energy fields, showing that there is no blow up. The pressure tensors, and the velocity fields constitute an exact solution to the 3D Navier-Stokes equation for a compressible fluid (Muriel, Results in Physics, 2011). All calculated fields are smooth, producing no turbulence by any generous definition of turbulence. We raise the question: is the Navier-Stokes equation the correct problem definition for turbulence? (Muriel, The Quantum Nature of Turbulence, Nova Science Publishers, New York (2010). We will display the plots of the solution in a 3D toroid configuration. The solution found may be used to reformulate fluid dynamics using an initial value formulation.

4:53PM L10.00007 Adaptive Control of the Forced Generalized Korteweg-de Vries-Burgers Equation

, NEJIB SMAOUI, ALAA EL-KADRI, MOHAMED ZRIBI, Kuwait University — The adaptive control problem of the forced generalized Korteweg-de Vries-Burgers (GKdVB) equation when the spatial domain is $[0, 1]$ is considered. Three different adaptive control laws are designed for the forced GKdVB equation. The L^2-global exponential stability of the solutions of these equations is shown for each of the proposed control laws by using the Lyapunov theory. Numerical simulations based on the Finite Element method (FEM) are presented to validate the analytical developments.

5:06PM L10.00008 Phase space dynamical density functional theory for colloids with hydrodynamic interactions

, BENJAMIN GODDARD, ANDREAS NOLD, PETR YATSYSHIN, NIKOS SAVVA, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK, GREG PAVLIOTIS, Department of Mathematics, Imperial College London, London SW7 2AZ, UK, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK — We study the dynamics of a colloidal fluid in the full position- momentum phase space. These dynamics are modelled by stochastic equations of motion for a large number of identical spherical particles. We include the full hydrodynamic interactions, which strongly influence the non-equilibrium properties of the system. For large systems, the number of degrees of freedom prohibits a direct solution of the equations and a reduced model is necessary. Under certain assumptions, we derive a dynamical density functional theory (DDFT), i.e. a reduction to the dynamics of the reduced one-body distribution. Our formulation includes the case where the momentum distribution is not a local Maxwellian. Near equilibrium, it reduces to a Navier-Stokes-like equation with additional non-local terms. In the high friction limit, we show rigorously that it reduces to a previously-derived DDFT, describing only the position distribution, but with a novel definition of the diffusion tensor.

5:19PM L10.00009 Attitudes Towards Fluids: the Impact of Flow Visualization

, JEAN HERTZBERG, University of Colorado, Boulder — Since 2003, a course on flow visualization has been offered to mixed teams of engineering and fine arts photography students at the University of Colorado. A survey instrument has been developed that explores student perceptions of and attitude towards fluid physics. It has been administered to students in the flow visualization course, in a traditional junior level fluid mechanics course, and in a course on design. Survey results indicate that the students in the flow visualization course, after a semester of making images for art's sake, emerge believing that fluid mechanics is more important to themselves as engineers and to society than the students in the traditional course which is packed with real-life engineering examples . The use of photography in improving student perceptions is being extended to a course on perception of design; preliminary results from a survey on attitudes towards design will be presented. Examples of student images from both courses will be presented as well.

5:32PM L10.00010 Applying the results of education research to help students learn more

, RACHEL E. PEPPER, STEPHANIE CHASTEEN, MICHAEL DUBSON, KATHERINE PERKINS, STEVEN POLLOCK, University of Colorado at Boulder — Over the past 5 years, the physics faculty at the University of Colorado have worked to transform three core courses in our upper-division undergraduate physics curriculum: Classical Mechanics/Math Methods, Electricity and Magnetism and Quantum Mechanics. We discuss our transformations as a potential model for transformation of other upper-division courses, such as fluid mechanics. The goal of our transformations was to improve student learning and to develop materials and approaches that other faculty could adopt or adapt. This work began with faculty in the department meeting regularly to define explicit course learning goals, which then served as a foundation for the subsequent course transformations. The development of the curriculum was also guided by the results of observations, interviews, and analysis of student work. We applied the principles of active engagement and learning theory to transform many elements of the course. Reforms included "clicker" questions, tutorials, modified homeworks, and more. In this talk, we will outline the process, the reforms, and present evidence of the effectiveness of these reforms relative to traditional courses. Some research-based fluid mechanics instructional materials will also be discussed. Our curriculum materials are available at http://www.colorado.edu/sei/departments/physics.htm.

Monday, November 21, 2011 3:35PM - 5:45PM –

Session L11 Turbulence Theory III 314

3:35PM L11.00001 Effects of Outer Scales on Near-Wall Reynolds Stresses and Higher-Order Statistics

, M.H. BUSCHMANN, Universitat Dresden, M. GAD-EL-HAK, Virginia Commonwealth University — The classical view of wall-bounded turbulence suggests that the near-wall region should be scaled with characteristic scales that are closely related to that region. For the last decade, however, alternative concepts considering the influence of outer scales were proposed. Herein, we show that the near-wall Reynolds stresses as well as higher-order statistics in different geometries (e.g., zero-pressure-gradient boundary layers, and pipe and channel flows) collapse in single Reynolds-number-independent curve when scaled with an alternative mixed scaling based on $\mathbf{u_\tau^{3/2} u_e^{1/2}}$.

3:48PM L11.00002 Relevant length scales and time scales in shear flow turbulence

, SUBHAS VENAYAGAMOORTHY, LAKSHMI DASI, BENJAMIN MATER, Colorado State University — Shear flow turbulence has been the subject of fundamental research due to its ubiquitous presence in engineering and natural flows. In this study, we take a fresh approach using dimensional arguments tempered by physical reasoning to gain further insights on their phenomenology. Beginning with the four basic quantities: turbulent kinetic energy k, dissipation rate ϵ, kinematic viscosity ν and mean shear S, we construct six length scales and two time scales that are most relevant to this classical problem and discuss their implications on phenomenology. Analysis of the variation of all six length scales and two timescales using high-resolution DNS data of turbulent channel flow and homogeneous shear flow are used to highlight important transitions in the flow dynamics and provide a framework to explain the energy cascade process.

4:01PM L11.00003 On the smallest sub-kolmogorov mean length scale and its implications on phenomenology

, LAKSHMI DASI, SUBHAS VENAYAGAMOORTHY, Colorado State University, Fort Collins CO — The smallest scale in turbulence has been predicted to be less than the Kolmogorov scale η by a factor of $\mathrm{Re}^{1/4}$ and attributed to the intermittency of the turbulent kinetic energy dissipation rate ε. We show through dimensional arguments that this smallest limit corresponds to a new mean length scale based on turbulent kinetic energy k and kinematic viscosity ν, given by $(\nu^2/k)^{1/2}$. The independence of this scale with ε raises the issue of physical dependence of length scales and challenges classical phenomenology. Thus the notion that the smallest scales are set by the intermittent fluctuations of dissipation rate may be physically in-accurate. Given that ε is also set independently by the large scale and the turbulent kinetic energy, the physical consequence is that the dissipative portion of energy cascade is constrained between η and $(\nu^2/k)^{1/2}$. Another important implication stems from the fact that $(\nu^2/k)^{1/2}$ is a mean length scale. This alludes to the existence of even smaller scales of motion in the instantaneous field that are governed by the fluctuations in turbulent kinetic energy.

4:14PM L11.00004 Length scales in anisotropic turbulence

, GREGORY BEWLEY, KELKEN CHANG, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization — In isotropic turbulence, a single scalar function fully describes the velocity correlation tensor. The characteristic scales of this correlation function, the Taylor scale and the integral scale, then have an unambiguous interpretation. The integral scale, for example, is a measure of the most energetic scale of turbulence. Anisotropic turbulence is more complicated. We examined theoretically and experimentally the relationships between correlation functions measured in two directions in anisotropic turbulence. We found that the ratio of characteristic scales measured in the different directions was a function of the ratio of fluctuating velocities in the two directions. In the case of the integral scale, the inertial range scaling law controls the relationship. In other words, not only is the integral scale a measure of the large scale, but it is also connected to inertial range dynamics.

4:27PM L11.00005 Stirring turbulence with turbulence , WILLEM VAN DE WATER, HAKKI ERGUN CEKLI, RENE JOOSTEN, Physics Department, Eindhoven University of Technology, Postbus 513, 5600 MB Eindhoven, the Netherlands — We stir wind–tunnel turbulence with an active grid that consists of rods with attached vanes. The time–varying angle of these rods is controlled by random numbers. We study the response of turbulence on the statistical properties of these random numbers. The random numbers are generated by the Gledzer–Ohkitani–Yamada shell model, which is a simple dynamical model of turbulence that produces a velocity field displaying inertial–range scaling behavior. The range of scales can be adjusted by selection of shells. We find that the largest energy input and the smallest anisotropy are reached when the time scale of the random numbers matches that of the large eddies in the wind–tunnel turbulence. A large mismatch of these times creates a flow with interesting statistics, but it is not turbulence.

4:40PM L11.00006 PIV study of turbulence generated by fractal grids in a water tunnel[1] , RAFAEL FERNANDES, Imperial College London, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton, CHRISTOS VASSILICOS, Imperial College London — An experimental study of turbulence generated by low-blockage space-filling fractal square grids was performed using 2D Particle Image Velocimetry (PIV) in a water tunnel. In addition to the experimental technique (PIV) and the fact that it was carried out in water, this study has also the particularity of having considerable incoming free stream turbulence with an intensity, in the streamwise (u'/U) and spanwise (v'/U) directions, of 2.8 and 4.4 % respectively. Results on turbulence intensity and Taylor microscale of the flow generated by our fractal grids are in good agreement with the previous wind tunnel study of Mazellier and Vassilicos ["Turbulence without the Richardson-Kolmogorov cascade," *Phys. Fluids* **22**, 075101 (2010)] provided that a different normalization scale is used which takes into account the free stream turbulence characteristics. This normalisation scale is a good estimator of the turbulence peak downstream of not only fractal but also regular grids. Finally, local isotropy of fractal generated turbulence was checked based on the gradients estimated from the 2D velocity field and compared with the ones from regular grids.

[1]EPSRC Grant EP/H030875/1

4:53PM L11.00007 Extraction of nonequilibrium -7/3 energy spectrum in experimental measurement turbulence data , KIYOSI HORIUTI, Tokyo Institute of Technology, Japan, YUICHI MASUDA, Tokyo Institute of Technology — A perturbation expansion for the energy spectrum about a base Kolmogorov $k^{-\frac{5}{3}}$ steady state yields an additional $-7/3$ power component which is induced by the fluctuation of the dissipation rate ε and represents a nonequilibrium state. DNS study revealed actual existence of $-7/3$ spectrum in homogeneous shear flow at $Re_\lambda \approx 150$ (*Phys. Fluids* **23**, 035107 (2011)). In this study, an attempt is made to extract the same spectrum in the data measured using the hot wire anemometer in the experiment of the driving mixing layer, in which nearly constant mean shear is established ($Re_\lambda \approx 468$, Y. Tsuji (2008)). The time series data are converted to the spatially evolving data employing the Taylor's hypothesis and ε is obtained. To be in accordance with the statistical theory, the variations of ε in the time scale comparable to the integral length scale are considered. Nonequilibrium component is extracted applying a conditional sampling on $d\varepsilon/dt$, and it is shown that the deviation from the base $-5/3$ spectrum fits the $-7/3$ power slope. The temporal development of the spectrum is divided into two regimes, Phases 1 and 2. Large energy contained in the low- wavenumber range in Phase 1 is cascaded to the small scales in Phase 2. This energy transfer is accomplished by the reversal in the sign of -7/3 power component. These results agree well with the DNS.

5:06PM L11.00008 Concentration dependence of the effects of polymer additives on bulk turbulence[1] , HENG-DONG XI, HAITAO XU, EBERHARD BODENSCHATZ, Max-Planck Institute for Dynamics and Self-Organization — We present an experimental study of the polymer concentration dependence of the effects of minute long chain polymer additives on bulk turbulent flow. It is found that the measured Eulerian structure function of the velocity field is strongly modified by the presence of the polymer additives. And there exists a critical concentration below which only small scales are modified while above which both small scales and large scales are modified. We found that the critical concentration depends on the energy dissipation rate of the flow, this dependence can be explained by de Gennes' elastic theory on turbulence of dilute polymeric solution.

[1]This work is supported by the Max Planck Society, the Humboldt Foundation, and the German Research Foundation (DFG).

5:19PM L11.00009 What correlations between velocity differences and velocity sums tell us about small scale universality , GREG VOTH, Wesleyan University — An ongoing question about turbulent flows is the degree to which the small scales are universal because they become independent of the details of the forcing at large scales. Recent work has explored these issues by measuring correlations between velocity differences over a distance r (whose variance is dominated by scales near r) and velocity sums over the same distance (whose variance is dominated by the large scales). Some correlations between velocity differences and sums are required by the Navier-Stokes equations (Hosokawa, Prog. Theor. Phys. Lett., 118:169, 2007.) This talk will look at experimental measurements of correlations between velocity differences and velocity sums from several flows. The correlations which are required by Navier-Stokes dynamics do not appear to violate the assumption of independence between the large and small scales. However, there are other correlations in the experimental data which can only be explained by dependence of the small scales on the details of the forcing of the flows. The variance of the velocity differences shows a strong conditional dependence on the velocity sum that is different in different flows. The anisotropy of the velocity differences also shows conditional dependence on the velocity sum.

5:32PM L11.00010 POD models for turbulent convection in rectangular cells , JORGE BAILON-CUBA, JOERG SCHUMACHER, Technische Universität Ilmenau — Two low-dimensional models (LDM) for turbulent convection in rectangular cells, based on the Galerkin projection of the Boussinesq equations onto a finite set of empirical eigenfunctions, are presented. The empirical eigenfunctions are obtained from a Proper Orthogonal Decomposition (POD) of the fields using the Snapshot Method. The first case is a three-dimensional cell in which a classical turbulent Rayleigh-Bénard flow evolves. The second case is based on two-dimensional DNS data of mixed convection in a cell with heated obstacles as well as in- and outlets of air. In both cases, a quadratic inhomogeneous coupled ODE system is obtained for the evolution of the modal amplitudes. The truncation to a finite number (a few hundred) of degrees of freedom, requires the additional implementation of an eddy viscosity-diffusivity to capture the missing dissipation of the small-scale modes. The magnitude of this additional dissipation mechanism is determined by requiring statistical stationarity and a total dissipation that corresponds with the original DNS data. We introduce a mode-dependent eddy viscosity-diffusivity, which turns out to reproduce the large-scale properties of the turbulent convection qualitatively well.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L12 Turbulence: Mixing II 315

3:35PM L12.00001 The Eddy-Diffusivity in Turbulent Two-Particle Dispersion , DAMIEN BENVENISTE, Physics, Johns Hopkins University, GREGORY EYINK, Applied Mathematics, Johns Hopkins University — R. H. Kraichnan (1966) and T. S. Lundgren (1981) derived a formula for the eddy-diffusivity in Richardson's theory of turbulent 2-particle dispersion:

$$\eta_{ij}(\mathbf{r},t) = \int_0^t ds \langle (u_i(\mathbf{x}+\mathbf{r},t) - u_i(\mathbf{x},t))(u_i(\mathbf{x}+\mathbf{r},t|s) - u_j(\mathbf{x},t|s)\rangle.$$

This formula involves the Lagrangian velocity field $\mathbf{u}(\mathbf{x},t|s)$ experienced at time $s < t$ by the fluid particle which is at point $\mathbf{x}$ at time t. Evaluating this formula requires tracking fluid particles backward in time, a difficult task with standard DNS. We compute the formula using the JHU Turbulence Database,[1] which stores the entire spacetime history of a 1024^3 DNS of homogeneous, isotropic turbulence at $Re_\lambda = 433$. We average over particle pairs started at many different initial positions $\mathbf{x}$ with initial separations $\mathbf{r}$. We obtain a time-dependent eddy-diffusivity $\eta_{ij}(\mathbf{r},t)$ which has Batchelor scaling $(\varepsilon r)^{2/3}t$ for short time and Richardson scaling $\varepsilon^{1/3}r^{4/3}$ for long time. Our resulting diffusion model describes both the Batchelor and Richardson regimes and also predicts new phenomena not yet seen in experiment or simulations.

[1]http://turbulence.pha.jhu.edu/

3:48PM L12.00002 Passive Scalar in 2D Turbulence , RORY CERBUS, WALTER GOLDBURG, University of Pittsburgh, Department of Physics and Astronomy — We examined the behavior of a passive scalar in a 2D turbulent flow and confirm the so-called Batchelor scaling. For large r, the second order structure function $S_2(r) = <(\theta(x+r)-\theta(x))^2> \sim \log(r)$. For small r, $S_2(r) \sim r^2$. The logarithmic dependence of $S_2(r)$ is consistent with a power spectrum that goes as the inverse power of k, the wavenumber. These experiments are performed using a falling soap film as the 2D turbulent system and various colored dyes for the passive scalar, which is injected at a point. The decaying turbulence is generated using a comb oriented perpendicular to the film. It does not appear to matter whether the dye is injected above or below the comb. The measurements were made in the Eulerian frame at a single point. Time is then replaced by distance using the Taylor frozen turbulence hypothesis. The structure function is determined from the correlation function, which is calculated using a photon correlation scheme. The passive scalar measurements are compared with the behavior of thickness fluctuations in the soap film, which is another random variable.

4:01PM L12.00003 Encounter rates of Lagrangian particles in homogeneous isotropic turbulence , SATOSHI YOKOJIMA, Department of Systems Engineering, Shizuoka University, Japan, TAKASHI MASHIKO, TAKAHIRO MATSUZAKA, TAKASHI MIYAHARA — Contact rates of Lagrangian particles are investigated numerically by direct simulation of homogeneous isotropic turbulence. The flow Reynolds number, the number of particles, and the contact radius are systematically changed, and the effects on the contact rates are discussed. In the talk, results based on a kinematic simulation of turbulence by unsteady random Fourier modes will be also presented.

4:14PM L12.00004 Bounded Stochastic Shell Mixing Model: Further Development and Application to Inhomogeneous Scalar Mixing , T. VAITHIANATHAN, YANJUN XIA, LANCE R. COLLINS, Cornell University — Xia and Collins [*Physics of Fluids* **23** (6):065107, 2011] developed the Bounded Stochastic Shell Mixing (BSSM) model that takes into account the multi-scale nature of the turbulent mixing process. They successfully applied the model to mixing of isotropic scalars with an initial double-delta probability density function (PDF). To enforce the scalar bounds, they introduced a novel "zeroth mode" that precisely cancels the inherently non-conservative random terms in the formulation. The extension of the model to the mixing of inhomogeneous scalar fields uses notional particles that move with a fluctuating velocity that is chosen to conform with the underlying turbulent energy spectrum. A consistency condition further requires the particle motion in the direction of the mean scalar gradient be carefully connected to the generation of the scalar fluctuation. The appropriate constraint has been derived and is enforced by the numerical algorithm. This new formulation has been applied to turbulent mixing of a scalar slab of specified thickness. (In the limit of zero thickness, this reduces to the classical "line source" problem.) We analyze multiple scalars so that differential diffusion can be considered as well as the effect of the thickness of the slab (relative to the turbulence length scales). The predictions of the BSSM model compare well with direct numerical simulations.

4:27PM L12.00005 Variations in scalar transport characteristics between forwards and backwards turbulent dispersion[1] , CHIRANTH SRINIVASAN, DIMITRIOS PAPAVASSILIOU, The University of Oklahoma — Recent studies about backwards dispersion in time have shown differences with forwards turbulent dispersion, and have highlighted the importance of understanding these differences for practical applications. This work used a direct numerical simulation combined with a Lagrangian scalar tracking approach to obtain single particle and relative dispersion statistics for both forwards and backwards dispersion in an infinitely long turbulent channel flow. The computational domain was 4πh $\times$ 2h $\times$ 2πh in x, y, z (where h is the half channel height and h = 300 and h = 150). Results showed differences in the rates of forwards and backwards relative scalar dispersion. The variation in the rates of relative scalar dispersion for a variety of Prandtl numbers (Pr) from 0.1 to 50 was also investigated. The primary direction of heat transport was found to be oriented almost close to the direction perpendicular to the channel walls, for all regions of the channel and for Pr up to 1000. Higher and enduring rates of heat transport were observed for the case of backwards turbulent dispersion. In analogy with the physics of optics, a quantity named the "turbulent dispersive ratio" was introduced to indicate the differences between backwards and forwards dispersion.

[1]NSF CBET-0651180 and TeraGrid under TRAC TG-CTS090025

4:40PM L12.00006 Passive Scalar Mixing in Compressible Isotropic Turbulence[1] , ADAM J. WACHTOR, FERNANDO F. GRINSTEIN, Los Alamos National Laboratory, C. RICHARD DEVORE, Naval Research Laboratory — Turbulent mixing of a passive scalar is studied through use of implicit large-eddy simulation (ILES) in the context of forced compressible, isotropic turbulence with a mean scalar gradient. The under-resolved prediction of mixing by an under-resolved turbulent velocity field is the problem that the ILES framework addresses without using any explicit model. Low wavenumber forcing is done separately for the solenoidal and dilatational components of the velocity in order for the flow field to achieve a statistically stationary state. The efficiency of ILES allows for the creation of large time-volume ensembles at relatively low computational cost. Effects of Mach number, grid resolution, and the forcing ratio of solenoidal to dilatational kinetic energy on the flow and subsequent scalar mixing will be presented.

[1]Funded by LANL LDRD-ER on "LES Modeling for Predictive Simulations of Material Mixing."

4:53PM L12.00007 Stochastic transport models for mixing in variable-density turbulence , J. BAKOSI, J.R. RISTORCELLI, Los Alamos National Laboratory — In variable-density (VD) turbulent mixing, where very-different- density materials coexist, the density fluctuations can be an order of magnitude larger than their mean. Density fluctuations are non-negligible in the inertia terms of the Navier-Stokes equation which has both quadratic and cubic nonlinearities. Very different mixing rates of different materials give rise to large differential accelerations and some fundamentally new physics that is not seen in constant-density turbulence. In VD flows material mixing is active in a sense far stronger than that applied in the Boussinesq approximation of buoyantly-driven flows: the mass fraction fluctuations are coupled to each other and to the fluid momentum. Statistical modeling of VD mixing requires accounting for basic constraints that are not important in the small-density-fluctuation passive-scalar-mixing approximation: the unit-sum of mass fractions, bounded sample space, and the highly skewed nature of the probability densities become essential. We derive a transport equation for the joint probability of mass fractions, equivalent to a system of stochastic differential equations, that is consistent with VD mixing in multi-component turbulence and consistently reduces to passive scalar mixing in constant-density flows.

5:06PM L12.00008 Consistency and realizability requirements for stochastic diffusion models for variable-density turbulent mixing , J.R. RISTORCELLI, J. BAKOSI, Los Alamos National Laboratory — Rational turbulence model development has applied principles that ensure consistency with the physical conservation laws and statistical constraints. Examples are the principles of invariance and realizability (Lumley, Adv. Appl. Mech., 18, 1979, pp. 123–176), and linearity and independence of passive scalars in mixing (Pope, Phys. Fluids, 26, 1983, pp. 404–408). Models that violate these principles can produce unphysical results. We discuss modeling principles and constraints for variable-density multi-material turbulent mixing. We develop the consequences of the mass conservation law for multi-component mixtures for random-walk methods in variable-density turbulence. In such flows the density fluctuations can be larger than the mean density and several important constraints restrict the functional forms of mixing models. One consequence of the constraints developed is that the coefficient of the Wiener process (if nonzero) must be nonlinear and coupled to the other mass fractions to ensure consistency with mass conservation. Typical Langevin-type models for these processes violate these constraints peculiar to variable-density mixing.

5:19PM L12.00009 Modeling the subfilter scalar variance for large eddy simulation in forced isotropic turbulence , ADAM CHEMINET, Ecole Polytechnique, France, GUILLAUME BLANQUART, California Institue of Technology — Static and dynamic model for the subfilter scalar variance in homogeneous isotropic turbulence are investigated using direct numerical simulations (DNS) of a lineary forced passive scalar field. First, we introduce a new scalar forcing technique conditioned only on the scalar field which allows the fluctuating scalar field to reach a statistically stationary state. Statistical properties, including 2nd and 3rd statistical moments, spectra, and probability density functions of the scalar field have been analyzed. Using this technique, we performed constant density and variable density DNS of scalar mixing in isotropic turbulence. The results are used in an a-priori study of scalar variance models. Emphasis is placed on further studying the dynamic model introduced by G. Balarac, H. Pitsch and V. Raman [Phys. Fluids 20, (2008)]. Scalar variance models based on Bedford and Yeo's expansion are accurate for small filter width but errors arise in the inertial subrange. Results suggest that a constant coefficient computed from an assumed Kolmogorov spectrum is often sufficient to predict the subfilter scalar variance.

5:32PM L12.00010 Effect of initial conditions on mixing within a turbulent channel flow , EMMANUEL GERMAINE, LAURENT MYDLARSKI, LUCA CORTELEZZI, McGill University — We analyze the mixing of a passive scalar (temperature) in a turbulent channel flow for different initial conditions by means of numerical simulations. The numerical domain is a channel delimited by two parallel and infinite flat walls, simulated using periodic boundary conditions in the streamwise and spanwise directions. We consider three initial distributions of temperature, where hot and cold fluids are separated by a sharp but continuous interface that subdivides the computational domain into two identical halves. The interface is taken parallel to the walls or perpendicular to them, oriented in the streamwise or spanwise directions. We perform a direct numerical simulation of the temperature field at $Re_\tau = 190$ when the flow is fully turbulent. The numerical scheme combines an advection diffusion solver, i.e, a third-order flux integral method based on UTOPIA (Leonard *et al.*, Appl. Math. Modeling, 1995), with a Navier-Stokes solver, i.e, spectral code released by Prof. John Gibson, http://www.channelflow.org). We quantify the time-evolution of the mixing performance of the turbulent flow using different measures of the mixing, including a negative Sobolev norm – a diagnostic currently used to asses the mixing performance of laminar flows. Finally, we discuss the influence of the initial conditions on the turbulent mixing. Funding was provided by NSERC (grants RGPIN217169 and RGPIN217184).

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L13 Multiphase Flow V: Numerical II 316

3:35PM L13.00001 A second order Lagrangian Eulerian momentum bounded method for multiphase flows[1] , VINCENT LE CHENADEC, HEINZ PITSCH, Center for Turbulence Research, Stanford University — A Lagrangian Eulerian framework relying on both Level Set and Volume of Fluid methods is presented in the context of multiphase flow computations. The resulting interface capturing scheme is shown to preserve planarity, and to conserve mass exactly for solenoidal and linear velocity fields. A novel fractional step approach for the incompressible Navier Stokes equation is also presented. The proposed scheme relies on a consistent transport of volume fraction and momentum fields, which also preserves velocity boundedness. A sharp interface projection step is derived accordingly. The algorithm is shown to conserve momentum exactly for solenoidal linear velocity, and to lead to robust computations.

[1]Supported by NASA under Subsonic Fixed Wing project.

3:48PM L13.00002 A GPU-accelerated flow solver for incompressible two-phase fluid flows , STEPHEN CODYER, MEHDI RAESSI, GAURAV KHANNA, University of Massachusetts Dartmouth — We present a numerical solver for incompressible, immiscible, two-phase fluid flows that is accelerated by using Graphics Processing Units (GPUs). The Navier-Stokes equations are solved by the projection method, which involves solving a pressure Poisson problem at each time step. A second-order discretization of the Poisson problem leads to a sparse matrix with five and seven diagonals for two- and three-dimensional simulations, respectively. Running a serial linear algebra solver on a single CPU can take 50-99.9% of the total simulation time to solve the above system for pressure. To remove this bottleneck, we utilized the large parallelization capabilities of GPUs; we developed a linear algebra solver based on the conjugate gradient iterative method (CGIM) by using CUDA 4.0 libraries and compared its performance with CUSP, an open-source, GPU library for linear algebra. Compared to running the CGIM solver on a single CPU core, for a 2D case, our GPU solver yields speedups of up to 88x in solver time and 81x overall time on a single GPU card. In 3D cases, the speedups are up to 81x (solver) and 15x (overall). Speedup is faster at higher grid resolutions and our GPU solver outperforms CUSP. Current work examines the acceleration versus a parallel CGIM CPU solver.

4:01PM L13.00003 The Lagrangian filtered mass density function (LFMDF) or LES/PDF method for turbulent two-phase flows

, SERGIO CHIBBARO, Universite Pierre et Marie Curie Paris, JEAN-PIERRE MINIER, EDF R&D Chatou — In this talk, a new formalism for the filtered density function (FDF) approach is developed for the treatment of turbulent polydispersed two-phase flows in LES simulations. Contrary to the FDF used for turbulent reactive single-phase flows, the present formalism is based on Lagrangian quantities and, in particular, on the Lagrangian filtered mass density function (LFMDF) as the central concept. This framework allows modeling and simulation of particle flows for LES to be set in a rigorous context and various links with other approaches to be made. In particular, the relation between LES for particle simulations of single-phase flows and Smoothed Particle Hydrodynamics (SPH) is put forward. Then, the discussion and derivation of possible subgrid stochastic models used for Lagrangian models in two-phase flows can set in a clear probabilistic equivalence with the corresponding LFMDF. Finally, a first stochastic model will be proposed in this framework and numerical simulations will show the comparison of LES simulations against DNS.

4:14PM L13.00004 An extended quadrature method of moments for polydisperse multiphase flows[1]

, CANSHENG YUAN, RODNEY FOX, DEPARTMENT OF CHEMICAL AND BIOLOGICAL ENGINEERING, IOWA STATE UNIVERSITY, AMES, IA, USA TEAM — Polydisperse multiphase flows arise in many applications, and thus there has been considerable interest in the development of numerical methods to find solutions to the kinetic equation used to model such flows. Quadrature-based moment methods (QBMM) are an important class of methods for which the accuracy of solution can be improved in a controlled manner by increasing the number of nodes. However, when large numbers of nodes are required to achieve the desired accuracy, the moment-inversion problem can become ill-conditioned. In this work, a new generation of quadrature algorithms is introduced that uses an explicit form for the distribution function. This extended quadrature method of moments (EQMOM) approximates the distribution function by a sum of classical weight functions, which allow unclosed source terms to be computed with great accuracy by increasing the number of quadrature nodes independent of the number of transported moments. Here we use EQMOM to solve a kinetic equation with evaporation, aggregation and breakage terms and compare the results with analytical solutions.

[1]This work was supported by a Grant from the US National Science Foundation (CCF-0830214).

4:27PM L13.00005 Realizable High-Order Finite-Volume Schemes for Diffusion in Quadrature-Based Moment Methods

, RODNEY O. FOX, VARUN VIKAS, Z.J. WANG, Iowa State University — Population balance equations (PBEs) can be reformulated in terms of the moments of the distribution function and a quadrature-based moment method (QBMM) can be used to solve them. The success of the QBMM is based on a moment-inversion algorithm that does not work if the moments are non-realizable. For convection terms, the authors have shown that when using a finite-volume approach, a moment-based cellwise reconstruction may lead to non-realizable schemes and hence a reconstruction based on weights and abscissas should be used instead. However, researchers working with diffusive PBEs have not reported realizability problems when using cellwise moment-based reconstruction. This work shows that when moment-based reconstruction with a 2^{nd}-order finite-volume scheme is used, realizability is automatically guaranteed by the satisfaction of Courant-Friedrichs-Lewy (CFL) condition. However, for any high-order finite-volume schemes, a moment-based reconstruction may fail to guarantee realizability. We present high-order realizable schemes based on reconstruction of weights and abscissas. These new schemes give a better performance for a certain class of diffusive PBE problems. Realizability conditions are also presented for a general unstructured mesh.

4:40PM L13.00006 DNS of droplet-laden incompressible turbulence: surface tension in a VoF method

, ALBERTO BARALDI, ANTONINO FERRANTE, University of Washington, Seattle — We investigated the continuous surface force (CSF) model to include the surface tension within a split-advection and mass-conserving volume of fluid (VoF) method for DNS of droplet-laden incompressible turbulence. The liquid-gas interface curvature is computed accurately using a variable-stencil height-function technique. Different implementations of the surface tension and pressure gradient terms within a projection method were tested, and their stability evaluated in terms of the magnitude of spurious currents for a droplet at rest in both two and three dimensions. The inherent asymmetry of the split-advection algorithm is reflected in the results of this test case. Our results show that a machine-accurate balance between pressure and surface tension forces can be achieved by enforcing symmetry of the VoF function. We have modified the sequence of the advection sweeps, and our results show that, in the case of non-zero Weber number, e.g. when a mean droplet velocity is present, the algorithm is accurate and stable. We present DNS results of fully-resolved droplet-laden incompressible isotropic turbulence.

4:53PM L13.00007 A Multiscale Approach to Compute Mass Transfer in Bubbly Flows[1]

, BAHMAN ABOULHASANZADEH, GRETAR TRYGGVASON, University of Notre Dame — Mass transfer in the liquid phase of gas-liquid multiphase flows usually takes place at a considerably slower rate than the transfer of momentum, so mass flux boundary layers are much thinner than momentum boundary layers. In Direct Numerical Simulations (DNS) the resolution requirement for flows where the Schmidt number (kinematic viscosity divided by mass diffusivity) is high are therefore significantly higher than for flow without mass transfer. While it is, in principle, possible to capture the mass transfer using adaptive grid refinement, the structure of the boundary layer is relatively simple and well understood. Here we develop a multi- scale approach to compute the mass transfer from buoyant bubbles, using a boundary-layer approximation next to the bubble and a relatively coarse grid for the rest of the flow. We show that the approach works well by comparing the results both with fully resolved simulations for modest Schmidt number and with empirical correlations for high Schmidt numbers.

[1]Research supported by NSF.

5:06PM L13.00008 ABSTRACT WITHDRAWN —

5:19PM L13.00009 ABSTRACT WITHDRAWN —

5:32PM L13.00010 An improved connectivity-free point set method to simulate multiphase flow

, CHU WANG, LUCY ZHANG, Rensselaer Polytechnic Institute — An improved connectivity-free point set method is presented to simulate the multiphase flow. Similar to the front tracking method, the point-set method tracks the interface explicitly. However, it does not require any logical connectivity among interface markers, which provides the flexibility in modeling large morphological changes such as bubble merging and breaking up. The topology changes are handled automatically by proper interface reconstruction scheme and also conveniently ease the small interface undulation due to advection. A meshfree RKPM interpolation method is employed to improved the algorithm which can handle non-uniform meshes and provide boundary corrections for free surface flows in situations when interface markers end at walls. Great accuracy is achieved for both the unit normal and curvature calculation. The incompressible two-phase flow is simulated using stabilized finite element method. Several test cases are performed to validate the improved method and show its capability in simulating multiphase type of flows successfully.

Monday, November 21, 2011 3:35PM - 5:32PM —

Session L14 Experimental Techniques III

3:35PM L14.00001 An optimal arrangement of a three or four hot-wire sensor array to simultaneously measure velocity component statistics in turbulent wall flows , JAMES WALLACE, University of Maryland, PETAR VUKOSLAVCEVIC', University of Montenegro — A highly resolved turbulent channel flow DNS with $Re_\tau = 200$ has been used to investigate the ability of probes made up of arrays of three or four hot-wire sensors to simultaneously and accurately measure statistics of all three velocity components in turbulent wall flows. Such arrays have also been combined in probes to measure, in addition, velocity gradient based statistics. Various virtual sensor arrangements have been tested in order to study the effects of position, number of sensors and spatial resolution on the measurements. First, the effective cooling velocity was determined for each sensor of an idealized probe, where the influence of the velocity component tangential to the sensors and flow blockage by the presence of the probe are neglected. Then, simulating the response of the virtual probes to obtain the effective velocities cooling the sensors, velocity component statistics have been calculated neglecting the velocity gradients over the probe sensing area. A strong influence of both mean and fluctuation velocity gradients on measurement accuracy was found. A new three-sensor array configuration designed to minimize the influence of the velocity gradients is proposed, and its accuracy is compared to two-sensor X- and V-array configurations.

3:48PM L14.00002 An in-situ calibration technique for four-wire hot-wire probe in conjunction for atmospheric studies , REZA SADR, Assistant Professor, ARINDAM SINGHA, Postdoctoral Research Associate — There is an increasing need to resolve the small-scales of atmospheric turbulence in order to estimate the higher order statistics of the turbulent flow. Sonic anemometers are commonly used in atmospheric research; however their application can only provide data with low special and temporal resolution. Hot-wire (HW) probes are still the best tool to obtain turbulent statistics with high temporal and spatial resolution. But HW probes are rarely used for atmospheric measurement due to the intricacy and logistical difficulties associated with the calibration and applications of the required probes for this flow field. In the present study, an in-situ method of calibration of a four-wire hot-wire anemometer is proposed, which bypasses the need for prior calibration. A proper data reduction algorithm has been developed to be used in conjunction with the four-wire probe. The proposed methodology enables one to use the hot-wire anemometer for atmospheric measurement to obtain three dimensional velocity information, at high spatial and temporal resolution, without the necessity of going through extensive calibration procedure. The feasibility of this method has been tested in laboratory and Monte Carlo simulation has been used to establish the stability and sensitivity of the data reduction algorithm.

4:01PM L14.00003 New measurement technique for turbulent flow as a replacement for hot-wire anemometry , JAROSLAW PUCZYLOWSKI, MICHAEL HÖLLING, JOACHIM PEINKE, ForWind, Institute of Physics - University of Oldenburg — We present latest developments of the 2d-Laser Cantilever Anemometer (2d-LCA), a sensor, which has been developed for highly resolved measurements of turbulent flows. Its measuring principle allows for high temporal resolutions of beyond 50kHz at spatial scales in sub millimeter range. This performance is achieved by measuring the deformation of a tiny cantilever via laser pointer, which experiences drag forces caused by the flow. The cantilever features two deformation modes, i.e. bending and twisting, whereas the latter occurs only for oblique inflow. Thus the sensor is capable of measuring two velocity components. Latest developments focus on the design of the cantilever. For example, an additional structure for a better sensitivity towards cross winds and an improved reflection pad were realized. Further improvements concern the laser beam guiding within the sensor. Beside this we are in the process of setting up advanced electronics and new types of PSD-elements with the goal of increasing the sensitivity. Comparison measurements between the re-designed 2d-LCA and older versions were performed and showed improvements relating signal quality and reliability. Further measurements in turbulent flow with an x-wire as a reference confirmed the ability of the new sensor to carry out measurements at comparable high resolutions.

4:14PM L14.00004 Multi-color particle shadow accelerometry (cPSA)[1] , GRANT DOWELL, MCPHAIL MICHAEL, ARNOLD FONTAINE, MICHAEL KRANE, ARL Penn State, LARRY GOSS, Innovative Scientific Solutions, Inc., JAMES CRAFTON, Innovative Scientific Soultions, Inc. — We present an extension of multi-color particle shadow velocimetry (cPSV) to unsteady acceleration measurement. cPSV uses a multi-color, pulsed LED light source for illumination. Particle shadow images recorded by a digital color camera are color separated and inverted. Standard DPIV processing methods are then used to estimate 2-D displacement vector fields. Acceleration estimates are facilitated by acquisition of three sequential images (one per color) in each camera exposure. Here, we prove the technique by measuring the tangential acceleration of a moving solid body, and compare the results to accelerometer measurements. We also present preliminary acceleration measurements performed in a near-wall turbulent pipe flow.

[1]Acknowledge support of NIH grant 5R01DC005642 ARL undergraduate honors program

4:27PM L14.00005 Development of an Acoustic Localization Method for Cavitation Experiments in Reverberant Environments , MINNA RANJEVA, LEE THOMPSON, Penn State - ARL, DANIEL PERLITZ, Penn State - ARL, WILLIAM BONNESS, DEAN CAPONE, BRIAN ELBING, Penn State - ARL — Cavitation is a major concern for the US Navy since it can cause ship damage and produce unwanted noise. The ability to precisely locate cavitation onset in laboratory scale experiments is essential for proper design that will minimize this undesired phenomenon. Measuring the cavitation onset is more accurately determined acoustically than visually. However, if other parts of the model begin to cavitate prior to the component of interest the acoustic data is contaminated with spurious noise. Consequently, cavitation onset is widely determined by optically locating the event of interest. The current research effort aims at developing an acoustic localization scheme for reverberant environments such as water tunnels. Currently cavitation bubbles are being induced in a static water tank with a laser, allowing the localization techniques to be refined with the bubble at a known location. The source is located with the use of acoustic data collected with hydrophones and analyzed using signal processing techniques. To verify the accuracy of the acoustic scheme, the events are simultaneously monitored visually with the use of a high speed camera. Once refined testing will be conducted in a water tunnel. This research was sponsored by the Naval Engineering Education Center (NEEC).

4:40PM L14.00006 MLM: Dust Explosion Potential Warning System , JOHN FOSS, ALAN LAWRENZ, Michigan State University — A quite large range of materials, when dispersed as a dust cloud in air, can support an explosion. Empirically derived minimum explosive concentration (MEC) values are typically in the range: 30-80 grams/m^3; that is, nominally 2.5-8.3% of STP density. Currently, there is no field-deployable measurement system to determine the mass loading (grams/m^3) of airborne dust. Proof-of-concept measurements for the MSU Mass Loading Monitor (MLM) are reported. A charge of dusty air, ingested into a cylinder, is accelerated (a_p=ct) by a driving piston and partially (≈8%) discharged from the open end of the cylinder. The deformable control volume momentum equation can be integrated with respect to time to yield $\alpha_{()} - \beta_{()} = \gamma\rho_{()}$ where ( ) will indicate with (w) or without (w/o) dust. The pressure integral (α) and the shear integral (β) balance the momentum within the cylinder at the end of the smoke plus the integral of the momentum flux. The kinematic attributes of these terms are represented by γ. It will be shown how the mass loading ($\rho_w - \rho_{w/o}$) can be determined. A full length paper (The Mass Loading Monitor Fundamental Principles And Proof Of Concept) will be published in *Meas. Sci. and Tech.*

4:53PM L14.00007 Comparing the sphere anemometer to standard sensors for 2D wind measurements , HENDRIK HEISSELMANN, MICHAEL HOELLING, JOACHIM PEINKE, University of Oldenburg & ForWind — The cup anemometers commonly used for wind energy applications are fairly robust, but suffer from several drawbacks like their limited temporal resolution, a systematic overestimation of the wind speed in turbulent flows and the inability to measure the wind direction. While sonic anemometers can measure the wind vector at a higher temporal resolution, they are more fragile and significantly more expensive. Therefore, we propose the sphere anemometer as a robust and highly-resolving alternative to standard anemometers. Designed without wearing parts, the sphere anemometer provides simultaneous wind speed and direction measurements as needed for wind turbine operation especially under challenging conditions such as offshore installation. In our contribution, we introduce the setup of the sphere anemometer which is based on the velocity-dependent deflection of a flexible tube with a sphere mounted atop. The deflection is measured in two dimensions using a light pointer, which allows for the simultaneous determination of wind speed and direction via calibration. Experimental results from wind tunnel measurements with sonic anemometer and sphere anemometer are presented, as well as first comparative measurements from the operation on the nacelle of a near-shore wind turbine.

5:06PM L14.00008 Geometrical Optimization of a Cylindrical Plasma Lens , BRIAN NEISWANDER, ERIC MATLIS, THOMAS CORKE, University of Notre Dame — Previous work by the authors have demonstrated the concept of an AC "plasma lens" for optical path difference (OPD) control of laser wavefronts. Plasma lenses feature no moving parts and a high frequency response, both of which are highly favorable for adaptive optics. This work investigates the geometrical constraints the plasma lens design, particularly the influence of the diameter of electrodes and gap distance between electrodes. A simplified electrostatic model for the plasma lens is developed and compared with experimental results. A good agreement is found between the experiments and theory. In regard to plasma lens design, the findings indicate (1) that there exists a critical gap-distance-to-diameter ratio, and (2) that the spatial efficiency of the device depends highly on the gap-distance-to-diameter ratio.

5:19PM L14.00009 Time-resolved tomographic PIV measurements in swirling jets , ANDREA IANIRO, Dipartimento di Ingegneria Aerospaziale (DIAS), University of Naples Federico II, Naples 80125, Italy, DANIELE VIOLATO, Aerospace Engineering Department, Delft University of Technology, Delft, The Netherlands, GENNARO CARDONE, Dipartimento di Ingegneria Aerospaziale (DIAS), University of Naples Federico II, Naples 80125, Italy, FULVIO SCARANO, Aerospace Engineering Department, Delft University of Technology, Delft, The Netherlands — The vortex dynamics of swirling jets at a Reynolds number of 1,000 is investigated by time-resolved tomographic Particle Image Velocimetry. Experiments are conducted at several values of the swirl number S from 0 to 0.8, therefore spanning the transition between weak and strong swirl regimes. Time resolved measurements are performed with the intention to shed more insight into the relation between shear layer instability at the outer side of the jet and the unsteady behaviour of the core region characterized by tumbling/precession motion. The average flow topology is addressed first in order to highlight the topological differences between weak and strong swirl flows. The three-dimensional topology of vortex structures is then visualized with Q criterion. Finally, the measured velocity and vorticity field is employed to estimate the terms of the vorticity equation, which enable to discuss on a more quantitative basis the effect on vortex dynamics and on the overall phenomenology of swirling jets.

Monday, November 21, 2011 3:35PM - 5:32PM –
Session L16 Separated Flows II 319

3:35PM L16.00001 Effect of wakes from moving upstream rods on boundary layer separation from a high lift airfoil[1] , RALPH J. VOLINO, U. S. Naval Academy — Highly loaded airfoils in turbines allow power generation using fewer airfoils. High loading, however, can cause boundary layer separation, resulting in reduced lift and increased aerodynamic loss. Separation is affected by the interaction between rotating blades and stationary vanes. Wakes from upstream vanes periodically impinge on downstream blades, and can reduce separation. The wakes include elevated turbulence, which can induce transition, and a velocity deficit, which results in an impinging flow on the blade surface known as a "negative jet." In the present study, flow through a linear cascade of very high lift airfoils is studied experimentally. Wakes are produced with moving rods which cut through the flow upstream of the airfoils, simulating the effect of upstream vanes. Pressure and velocity fields are documented. Wake spacing and velocity are varied. At low Reynolds numbers without wakes, the boundary layer separates and does not reattach. At high wake passing frequencies separation is largely suppressed. At lower frequencies, ensemble averaged velocity results show intermittent separation and reattachment during the wake passing cycle.

[1]Supported by NASA.

3:48PM L16.00002 Experimental investigation of frequency lock-on in separated flow[1] , JOHN GRIFFIN, LOU CATTAFESTA, University of Florida, JONATHAN TU, CLANCY ROWLEY, Princeton University, EHSAN ARAM, RAJAT MITTAL, Johns Hopkins University — Separated flow is a complex phenomenon comprised of several flow dynamics. Recent experimental and computational investigations propose that, depending on flow conditions, the nature of flow separation is governed by three dominant mechanisms: the shear layer, separation bubble, and wake instabilities. The nonlinear interactions of these instabilities provide potential for separation characterized by various lock-on states. This study provides an experimental investigation into the lock-on type dynamics for separated flows from various two-dimensional airfoil shapes, Reynolds numbers, and angles of attack. Two simultaneously sampled hot-wire probes are used to acquire velocity in two of the three regions of interest. The data quantify nonlinear coupling between the instabilities observed in the shear layer, the wake, and, in the case of mean flow reattachment, the separation bubble. The locations of these phenomena are determined with simple two-component particle image velocimetry. The coupling is assessed via higher-order spectral and moment analysis. Several of the cases that demonstrate resonant behavior are included for discussion.

[1]Supported by AFOSR grant FA9550-09-1-0257.

4:01PM L16.00003 Wake Modes of Rotationally Oscillating Cylinders at low Re , PRABU SELLAPPAN, TAIT POTTEBAUM, University of Southern California — Vortex shedding from bluff bodies is important in various engineering applications because the wake can have many effects, including exciting vibrations in structures and altering convective heat transfer. While vortex shedding from cylinders in cross-flow and cylinders undergoing transverse and in-line oscillations has been studied extensively, only limited data is available for rotational oscillations and is mainly limited to spectral analysis of the wake. Water tunnel experiments were carried out at Re = 150 to investigate the wake of a rotationally oscillating cylinder for oscillation frequencies from 0.67 to 3.5 times the natural shedding frequency and peak-to-peak oscillation amplitudes up to 320°. DPIV was used to study both the near and far wake within this parameter space. Well-defined patterns of wake vortices were observed in distinct regions of the parameter space, similar to the wake modes of transversely oscillating cylinders in cross-flow. In portions of the parameter space for which information exists in the literature the wake modes are well-related to spectral data. Variants of modes in previously unexplored regions are explained in terms of harmonics. The initial application of these results to understanding heat transfer enhancement from rotationally oscillating cylinders will also be addressed.

4:14PM L16.00004 Separation Control in a Centrifugal Bend Using Plasma Actuators, MICHAEL ARTHUR, THOMAS CORKE, University of Notre Dame — An experiment and CFD simulation are presented to examine the use of plasma actuators to control flow separation in a 2-D channel with a 135° inside-bend that is intended to represent a centrifugal bend in a gas turbine engine. The design inlet conditions are $P = 330\,\text{psia.}$, $T = 1100°\text{F}$, and $M = 0.24$. For these conditions, the flow separates on the inside radius of the bend. A CFD simulation was used to determine the location of the flow separation, and the conditions (location and voltage) of a plasma actuator that was needed to keep the flow attached. The plasma actuator body force model used in the simulation was updated to include the effect of high-pressure operation. An experiment was used to validate the simulation and to further investigate the effect of inlet pressure and Mach number on the flow separation control. This involved a transient high-pressure blow-down facility. The flow field is documented using an array of static pressure taps in the channel outside-radius side wall, and a rake of total pressure probes at the exit of the bend. The results as well as the pressure effect on the plasma actuators are presented.

4:27PM L16.00005 An experimental study of flow past an oscillating cylinder, SANJAY KUMAR, CARLOS LOPEZ, GERMAN FRANCISCO, DAVOOD ASKARI, Department of Engineering, The University of Texas at Brownsville, CESAR ALEJANDRO TOLEDO SUAREZ, OLIVER PROBST, Physics Department, Instituto Tecnologico y de Estudios Superiores de Monterrey (Mexico) — We present preliminary experimental results on flow past an oscillating cylinder at frequency ratios varying from 0 to 5 and oscillation amplitudes varying from $\pi/8$ to π. The experiments are conducted at Reynolds number of 185. The frequency ratio, f_R, is defined as the ratio of cylinder oscillation frequency to vortex shedding frequency from a non-oscillating cylinder. The diagnostic is done using hydrogen bubble technique for flow visualization in a plane. It is found that at one diameter downstream from the cylinder, vortex shedding frequency matches the forcing frequency (lock-on) at all $f_R > 1$ and all amplitudes; however, for $f_R < 1$ there is a window adjacent to $f_R = 1$ where lock-on occurs and this depends on oscillation amplitude. In the far wake at nine diameters from the cylinder, the lock-on region is centered around $f_R = 1$ and depends strongly on amplitude. The visualization of the vortical structures showed that near $f_R = 1.0$ the vortices became very compact and well formed in the visualization plane. Their size decreases considerably at much higher f_R's in the near-wake. The motion pictures reveal interesting phenomenon of merger of vortices at certain f_R's.

4:40PM L16.00006 Experimental Study of Wake Instabilities of a Blunt Trailing Edge Profiled Body at Intermediate Reynolds Numbers[1], ARASH NAGHIB-LAHOUTI, PHILIPPE LAVOIE, University of Toronto, HORIA HANGAN, University of Western Ontario — The periodic shedding of von Kármán vortices is the primary instability in the wake of nominally 2D bluff bodies, beyond a critical Reynolds number around 45-49. When Reynolds number passes a second threshold, which can be as high as 700 depending on profile geometry, secondary instabilities emerge and accompany the von Kármán vortices. For most bluff bodies, these instabilities appear as pairs of counter-rotating streamwise vortices, and spanwise undulations of the von Kármán vortices. The mechanism and scale of these instabilities depend on the bluff body geometry and Reynolds number. The focus of the present study is to identify and characterize the dominant secondary instability in the wake of a blunt trailing edge profiled body at intermediate Reynolds numbers between 8,000 and 20,000 based on the body thickness. The experiments, which include PIV and hot-wire measurements in the wake, complement previous studies involving the same bluff body at higher and lower Reynolds numbers, and make it possible to determine the scale and mechanism of the secondary instability at intermediate Reynolds numbers.

[1]Funded partly by the Government of Ontario, MITACS and Bombardier Aerospace.

4:53PM L16.00007 Comparison of turbulent separation over a smooth surface and mako shark skin on a NACA 4412 hydrofoil[1], DREW SMITH, AMY LANG, REDHA WAHIDI, University of Alabama — Shark skin is being investigated as a means of passive flow separation control due to the flexibility and preferential flow direction of the scales covering the skin. In this study, the effect of the scales is observed in a tripped turbulent boundary layer by comparing the flow over a NACA 4412 hydrofoil with a smooth surface to that over the same hydrofoil with samples of mako shark skin affixed to its upper surface. These samples were taken from the flank area of the shark because the scales at that location have been shown to have the greatest angle of erection, and thus the best potential for separation control. All flow data in this study was obtained using Time-Resolved Digital Particle Image Velocimetry and recorded at multiple angles of attack (between 8 and 16 degrees) and two Reynolds numbers. The flow was primarily analyzed by means of the backflow coefficient (a value based on the percentage of time that flow in a region over the hydrofoil is reversed) and the time history of instantaneous flow velocity values at specific points in the boundary layer over the hydrofoil models.

[1]Research performed under NSF grant 0932352.

5:06PM L16.00008 An Experimental Study of Flow Separation over 2D Transverse Grooves, EMILY JONES, AMY LANG, FARHANA AFROZ, JENNIFER WHEELUS, DREW SMITH, The University of Alabama — A shark's scales help to reduce drag over its body by controlling boundary layer separation over its skin. It is theorized that the scales bristle when encountering a reversing flow, thereby trapping vortices between the scales, creating a partial slip condition over the surface and inducing turbulence augmentation in the boundary layer. In an attempt to replicate and study these effects, a spinning cylinder was used in a water tunnel to induce separation over a flat plate with 2 millimeter square 2D transverse grooves. The results were compared to separation occurring over a flat plate without grooves using DPIV. The angular speed of the cylinder was varied. The observed delays in separation, changes in separation bubble shedding frequency and other effects upon the boundary layer are discussed.

5:19PM L16.00009 Flow around the tip of a circular cylinder in proximity with a channel bottom wall, CHRISTOPHER HOCUT[1], University of Notre Dame, RALPH BUDWIG[2], University of Idaho — Measurement probes create a region of disturbed flow in the vicinity of the probe tip. The present investigation examined the disturbance due to an idealized probe (a circular cylinder) that penetrated the free surface of an open channel flow as well as potential interactions of the disturbance with the channel bottom. Stereoscopic particle image velocimetry (PIV) was used to measure the three components of velocity in the vertical plane of symmetry downstream the cylinder with the tip of the cylinder located zero to four diameters from the channel bottom. The disturbance characteristics (change in velocity and turbulence quantities), vertical distance to the extent of disturbance as measured from cylinder tip, and downstream distance to the point of maximum disturbance have been determined. The cylinder tip depth that caused interaction with the channel bottom leading to onset of scour was also investigated.

[1]Environmental Fluid Dynamics Laboratories
[2]Center for Ecohydraulics Research

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L17 Reacting Flows III: Engines, Sprays, and Soot 320

3:35PM L17.00001 The flow field in a rotating detonation-wave engine[1] , KAZHIKATHRA KAILASANATH, DOUGLAS SCHWER, Naval Research Laboratory — Rotating detonation-wave engines (RDE) are a form of continuous detonation-wave engine. They potentially provide further gains than an intermittent or pulsed detonation–wave engine (PDE). However, significantly less work has been on this concept when compared to the PDE. In this talk, we present the detailed flow field in an idealized RDE, primarily consisting of two concentric cylinders. A premixed detonable mixture is injected into the annulus between the two concentric cylinders. Once a detonation is initiated, it keeps travelling around in the annulus as long as there is fresh detonable mixture ahead of it. Hence, the injection process is critically important to the stability and performance of the RDE. Furthermore, we show that the flow field is quite complex consisting of multiple shock waves and the outflow is primarily axial, although the detonation-wave is travelling around circumferentially.

[1]Sponsored by the NRL 6.1 Computational Physics Task Area.

3:48PM L17.00002 Fluid dynamics in a Rotating-Detonation-Engine with micro-injectors[1] , DOUGLAS SCHWER, Naval Research Laboratory — Rotating detonation engines (RDE's) represent a natural extension of the extensively studied pulse detonation engines (PDE's) for obtaining propulsion from the high efficiency detonation cycle. RDE's require fuel and oxidizer under high pressure to be injected through micro-nozzles from one or two plenums (for premixed and non-premixed). This injection process is critically important to the stability and performance of the RDE. This paper studies the effect of this injection process on the detonation wave within the combustion chamber, with an emphasis on how the fluid dynamics are affected. Both two-dimensional and three-dimensional simulations are done using well proven numerical methods for both the combustion chamber and mixture plenums of an idealized RDE.

[1]This work is supported through NRL 6.1 Computational Physics Task Area

4:01PM L17.00003 LES Of A Model Aircraft Combustor: Spray Model Assumptions And Pollutant Formation[1] , EDWARD KNUDSEN, SHASHANK, Stanford University, HEINZ PITSCH, RWTH Aachen University — Large eddy simulations of NASA's Lean Direct Injection spray combustor are performed. Fuel spray is described using Lagrangian particles, and combustion is modeled using a multi-regime flamelet approach. Particular emphasis is placed on analyzing how spray model assumptions affect pollutant predictions. This analysis is performed by running a series of cases that employ different spray evaporation models, different spray cone angles, and different bulk air flow rates. Results demonstrate that both CO and NO are leading order sensitive to the details of the spray formulation. Temperature is also shown to be leading order sensitive to the spray formulation, while major species such as CO_2 and H_2O are less sensitive. These results highlight the difficulty of validating pollutant models with liquid fueled experiments that are subject to spray modeling uncertainty.

[1]Funding from NASA gratefully acknowledged.

4:14PM L17.00004 Theory of vaporization and combustion of fuel sprays in strained laminar mixing layers , JAVIER URZAY, Center for Turbulence Research, Stanford University, ANTONIO SANCHEZ, Departamento de Ingeneria Termica y de Fluidos, Universidad Carlos III de Madrid, HEINZ PITSCH, Institut fur Technische Verbrennung, RWTH Aachen University, AMABLE LINAN, ETSI Aeronauticos, Universidad Politecnica de Madrid — The vaporization and combustion of a monodisperse fuel spray in a laminar counterflow mixing layer is investigated under conditions such that the droplet Stokes number is smaller than 1/4, so that the droplets do not cross the stagnation plane, but instead tend to accumulate there because of their inertial slip motion. Vaporization is confined to the thin strained mixing layer separating the spray from the hot stream, which can be described with an Eulerian description for the liquid phase. The numerical integration of the resulting boundary-value problem provides the structure of the reactive mixing layer, including the standoff distance from the stagnation plane where droplets disappear and the flame location. Limiting asymptotic solutions for extreme values of the controlling parameters are also determined, and a generalized vaporization law for droplets in the presence of thermal gradients is derived. The results provide increased understanding of laminar spray flamelets.

4:27PM L17.00005 A multi-scalar PDF approach for LES of turbulent spray combustion , VENKAT RAMAN, COLIN HEYE, The University of Texas at Austin — A comprehensive joint-scalar probability density function (PDF) approach is proposed for large eddy simulation (LES) of turbulent spray combustion and tests are conducted to analyze the validity and modeling requirements. The PDF method has the advantage that the chemical source term appears closed but requires models for the small scale mixing process. A stable and consistent numerical algorithm for the LES/PDF approach is presented. To understand the modeling issues in the PDF method, direct numerical simulation of a spray flame at three different fuel droplet Stokes numbers and an equivalent gaseous flame are carried out. Assumptions in closing the subfilter conditional diffusion term in the filtered PDF transport equation are evaluated for various model forms. In addition, the validity of evaporation rate models in high Stokes number flows is analyzed.

4:40PM L17.00006 DNS of aerosol evolution in a turbulent jet , KUN ZHOU, ANTONIO ATTILI, FABRIZIO BISETTI, Reactive Flow Modeling Laboratory, King Abdullah University of Science and Technology — The effects of turbulence on the evolution of aerosols are not well understood. In this work, the interaction of aerosol dynamics and turbulence are studied in a canonical flow configuration by numerical means. The configuration consists of a hot nitrogen stream saturated with dibutyl phthalate (DBP) vapor mixing with cool air in a shear layer. A direct numerical simulation (DNS) for the momentum and scalar fields is coupled with the direct quadrature method of moments (DQMOM) for the condensing liquid phase. The effects of turbulent mixing on aerosol processes (nucleation, condensation, and coagulation) are quantified by analyzing the statistics of number density and droplet sizes.

4:53PM L17.00007 Soot formation in unstrained diffusion flames , ETIENNE ROBERT, Department of Mechanics, Kungliga Tekniska Högskolan (KTH), Stockholm, Sweden, NILS-ERIK OLOFSSON, JONATHAN JOHNSSON, HENRIK BLADH, PER-ERIK BENGTSSON, Combustion Physics, Lund University, Lund, Sweden — The formation of soot particles has been investigated in CH_4/O_2 diffusion flames using a burner which allows the creation of a nearly unstrained planar reaction sheet. The sooting limits, soot volume fraction and particle size were measured as a function of bulk flow across the flame mixture strength and transport properties of the reactants. Mass spectrometry was used to measure the effective mixture composition close to the flame and Laser Induced Incandescence (LII)for the soot volume fraction and particle size. The parameter space was mapped as follows: Starting from a stable non-sooting baseline flame, the mixture strength was progressively increased by raising the fuel volume fraction while keeping other parameters constant (bulk flow across the flame, oxidant and inert composition). As the mixture strength was increased, the soot volume fraction and particle size increased up to a point where very big soot particle aggregates became visible to the naked eye on the flame side of the sooting layer. The exact mechanism by which these super aggregates arise is unknown but it is postulated that the absence of strain in the flow field and the thermophoretic effect allows soot particles to remain in a region of the burning chamber suitable for growth for an extended period of time.

5:06PM L17.00008 Characterization of Mixing and Ignition Effects in Flow-Reactors Using a Particle Method , SIMON WEIHER, ETH Zurich, MATTHIAS IHME, University of Michigan — Recent investigations have indicated discrepancies between measurements and simulations of the ignition delay for syngas-mixtures at high-pressure/low- temperature conditions. While relevant sources for these discrepancies have been identified in the context of rapid compression machines and shock tubes, the underlying mechanisms and nonidealities in flow-reactor experiments have not yet been quantified. The objective of this investigation is to characterize effects of turbulence and flow-field inhomogeneities on the mixing and ignition-dynamics in flow-reactors. To this, an idealized flow-reactor is considered, in which the unsteady and three-dimensional flow-field is obtained from the solution of a large-eddy simulation. A particle method is used to describe the mixing, induction, and subsequent ignition of the reactants. Utilizing this model, parametric studies are performed to systematically quantify effects of initial mixture-preparation, flow-rate modulation, and turbulence-levels on the ignition-process over a range of practically relevant temperature and pressure conditions.

5:19PM L17.00009 Quantification of the uncertainties in the prediction of extinction of hydrogen-air diffusion flames , NICOLAS KSEIB, Mechanical Engineering Department, Stanford Unversity, JAVIER URZAY, GIANLUCA IACCARINO, Center for Turbulence Research, Stanford University — The study of the physical processes that lead to extinction of flames in gaseous hydrogen-air non-premixed combustion is of paramount importance for the reliable design of power plants and advanced propulsion systems in automobiles and hypersonic aircrafts. However, there remain several uncertainties in the experimental quantification of reaction rates of elementary steps in most of hydrogen-air mechanisms, which can produce hazards in hydrogen manipulation and engine malfunction. In this study, the effects of aleatory uncertainties in the chemical reaction-rate constants induced in hydrogen-air counterflow diffusion-flame extinction processes are addressed, with a probabilistic representation of the uncertain parameters sampled with a Markov-Chain Monte Carlo algorithm. Measurements of the reaction-rate constants and their associated uncertainty factors, reported earlier for the Stanford hydrogen-air detailed chemical mechanism, are used to study the propagation of uncertainties in the calculation of scalar dissipation rates at extinction. Non-intrusive methods are used to analyze the variablities, with the probability density function of the scalar dissipation rate being sampled around regions involving flame extinction and global sensitivity indices being computed by Monte Carlo sampling.

5:32PM L17.00010 Regression Rate Enhancement of Hybrid Rocket Motors using Mixed Hybrid Concept , PALANI KUMAR CHIDAMBARAM, AMIT KUMAR, IIT Madras, Chennai-600036, India — Low regression rates have been a major problem for hybrid rocket motors. In the present study, the effect on regression rate by adding ammonium perchlorate (AP) in solid fuel is studied numerically. AP mixed with HTPB is used as solid fuel and gaseous oxygen (GOX) is used as oxidizer. Solid fuel compositions are chosen such that the rocket motor retains start-stop capability. A reduced three step mechanism proposed in the literature is utilized to simulate the combustion. In the combustion chamber, two distinct flame fronts are captured. AP decomposition reaction forms a premixed flame front near the fuel surface. The AP decomposed products also react with HTPB. Heat released in these reactions improves the heat transferred to solid fuel and the regression rate significantly. Un-burnt fuel in the products further reacts with GOX forming a diffusion flame front farther from fuel surface. The presence of premixed flame front thus overcomes the low-regressing nature of hybrid combustion. It is found that 50% AP in solid fuel increases the regression rate by as much as 3 times.

Monday, November 21, 2011 3:35PM - 5:45PM – Session L18 Microfluids: Fluidic Devices I 321

3:35PM L18.00001 Levitation, aggregation and separation of micro-sized particles in a Hydrodynamic Acoustic Sorter, HAS , MAURICIO HOYOS, ANGELICA CASTRO, PMMH-CNRS-ESPCI, DESPINA BAZOU, Trinity College Dublin, SEPARATION COLLABORATION — Levitation, aggregation and separation of micron-sized particulate materials can be generated in a fluidic resonator by an ultrasonic standing wave field force. A piezoelectric transducer generates standing waves between the two walls of a parallel plate channel composing the resonator. The number of pressure nodes n is given by the relationship: $w = n\lambda/2$ with λ the wavelength. The primary radiation force generated by the standing wave generates levitation of micron-sized particles driving them toward the nodal planes. An equilibrium position is reached in the channel thickness where the acoustic force balances the gravity force. The equilibrium position is independent on particle size but it depends on the acoustic properties. Once particles reach the equilibrium position, transversal secondary forces generate aggregation. We shall present the levitation and aggregation process of latex particles and cancer cells in a 2MHz resonator. We demonstrate the possibility of separating particles under flow in a Hydrodynamic Acoustic Sorter HAS, in function of their acoustic impedance and in function of their size using a programming field force.

3:48PM L18.00002 Continuous Separation of Microparticles in a Microfluidic Channel via the Elasto-inertial Effect of Non-Newtonian Fluid , SEHYUN SHIN, JEONGHUN NAM, HYUNJUNG LIM, Korea University — Pure separation and sorting of microparticles from complex fluids are essential for biochemical analyses and clinical diagnostics. In this study, we present a simple and label-free method to separate microparticles with high purity using the elasto-inertial characteristic of non-Newtonian fluid in microchannel flow. At the inlet, particle-containing sample flow was pushed toward the side walls by introducing sheath fluid from the center inlet. Particles of 1 μm and 5 μm diameter that were suspended in viscoelastic fluid were successfully separated in the outlet channels: larger particles were notably focused on the centerline of the channel at the outlet, while smaller particles kept flowing along the sidewalls with minimal lateral migration to centerline. The same technique was further applied to separate platelets from diluted whole blood. Through cytometric analysis, we found that the purity of collected platelets was close to 99.9%. Conclusively, the technique of microparticle separation using elasto-inertial forces in non-Newtonian fluid is proven to be an effective method for separating and collecting microparticles based on size differences with high purity.

4:01PM L18.00003 Interactive tuning of flow geometry for size-sensitive sorting of microparticles , SASCHA HILGENFELDT, CHENG WANG, SHREYAS JALIKOP, University of Illinois at Urbana-Champaign — We show that sensitive selection, focusing, and sorting of microparticles by size is possible in microfluidic setups without the need for moving boundaries or external forces on the particles. The flow domain is flexibly and interactively shaped by the superposition of a transport flow and a microbubble-induced streaming flow. This method separates particles for which both the absolute size and the size differential are only a few micrometers, in a setup whose smallest geometric scale is about 100 microns. Size-dependent trapping, release, and focusing can be effected and used for switching and sorting [1]. Devices based on this novel concept are easy to fabricate and can be directly tailored to a variety of transported objects, including cells and vesicles.

[1] C. Wang, S. V. Jalikop and S. Hilgenfeldt, Applied Physics Letters **99**, 034101 (2011).

4:14PM L18.00004 Continuous size-based dielectrophoretic particle sorting in a microfluidic device , BARUKYAH SHAPARENKO, HAN-SHENG CHUANG, HOWARD HU, HAIM BAU, University of Pennsylvania — The dielectrophoresis (DEP) force acting on a particle passing through a nonuniform electric field is proportional to its volume, making DEP well-suited for size-based particle sorting. Pinched flow fractionation uses the geometry constraints of a narrow segment of microchannel to effect size-based separation. We combine these two techniques in series to create a size-based microfluidic sorting device, using negative DEP to allow for continuous particle sorting. An interdigitated array of five L-shaped electrodes permits the sorting of up to five different particle sizes. For a given set of particle sizes ($\sim$1–10 μm), this sorting process can be optimized by using the applied potentials on the electrodes as our optimization parameters. Through on-chip voltage control of the electrodes, we can achieve sorting for various sets of particle sizes with the same microfluidic device geometry. We compare the computational optimization solution to an analytical solution and with experimental results.

4:27PM L18.00005 Gravity driven separation based on lateral displacement in anisotropic microfluidic media , RAGHAVENDRA DEVENDRA, GERMAN DRAZER, Department of Chemical and Biomolecular Engineering, Johns Hopkins University, Baltimore MD 21218 — We designed an anisotropic periodic array of rectangular obstacles and directly measured the Brownian motion of particles in the absence of an external field. Earlier, we established that in the limit of small driving forces, particle-wall hydrodynamic interactions, coupled with geometric confinement, lead to non- linear effects in the mobility of rigid spherical particles in the periodic array. We further proposed that such anisotropic media could be used for continuous size-based separation of particles in microfluidic devices. Here, we show that such anisotropic systems could also lead to separation in the deterministic limit of high Peclet numbers. We use gravity as the driving force and investigate the effect of the orientation of the force with respect to the media. We observe directional locking in the motion of the particles, analogous to that observed in macroscopic systems and discuss the role of irreversible particle-obstacle interactions in the observed behavior. We report the average migration angles for different sizes of particles and confirm the utility of these anisotropic arrays to create microdevices for continuous particle sorting.

4:40PM L18.00006 Diamagnetic Particle Separation in Ferrofluid Microflows , XIANGCHUN XUAN, LITAO LIANG, Clemson University — Particle separation is important for a wide range of applications. A variety of force fields have been demonstrated to separate particles in microfluidic devices. Magnetic field-induced separation is simple, cheap, and free of fluid heating issues that accompany electric, acoustic, and optical methods. We develop a novel magnetic particle separation method in a curved microchannel with a nearby permanent magnet. This method is capable of separating both magnetic and nonmagnetic particles by size. It is based on the dependence of particle magnetophoresis on the particle size and the particle's distance from the magnet. We present in this talk a continuous separation of 3 μm- and 5 μm-diameter polystyrene particles in a ferrofluid flow without magnetic and fluorescent labeling. We also develop a numerical model to simulate the particle separation process.

4:53PM L18.00007 Continuous blood fractionation using an array of slanted grooves , JORGE A. BERNATE, CHEMBE JHU, LIU CHENGXUN, LIESBET LAGAE, IMEC - SSET/FNS, GERMAN DRAZER, CHEMBE JHU — Blood is a complex fluid having different specialized biological functions and containing a plethora of clinical information. The separation of different blood components is a crucial step in many research and clinical applications. In this work we take advantage of the flow characteristics in microfluidic devices in which the bottom surface is patterned with slanted rectangular grooves to continuously fractionate blood. We exploit the flow in the vicinity of the patterned surface when the dimensions of the grooves are much smaller than the dimensions of the main channel. In these devices, we observed that the grooves act as open channels guiding flow along them with the flow over them being in the direction of the main channel. We present experiments in which the different blood components are deflected laterally to a different extent by the flow along the grooves depending on their sedimentation velocity, which allows their continuous fractionation. In particular, the heavier red blood cells experience the largest deflection while the lighter white blood cells deflect the least, allowing their passive and minimally invasive isolation. In addition, this fluidic platform can also be used to separate magnetically labeled circulating cancer cells which can be retained in the flow along the grooves using a sufficiently strong magnetic force.

5:06PM L18.00008 Dielectrophoretic Separation of Live and Dead Yeast Cells in Microfluidic Reservoirs , DANIEL SHOWERS, VINCENT BROWN, LITAO LIANG, TZUEN-RONG J. TZENG, XIANGCHUN XUAN, Clemson University — Insulator-based dielectrophoresis (iDEP) is an emerging technology that has been widely used to manipulate particles and cells in microfluidic devices. Current iDEP devices use in-channel micro-obstacles such as hurdles, posts and ridges to create electric field fields, which may cause potential Joule heating problem due to the locally amplified electric field. In this talk we present a dielectrophoretic separation method in microfluidic reservoirs. Due to the significant size mismatch between a microchannel and its end-channel reservoirs, electric fields gradients are inherently produced at the microchannel-reservoir junction. The induced dielectrophoresis can be utilized to focus and trap cells and particles. We demonstrate a continuous concentration and a selective isolation of live yeast cells from dead yeast cells in a reservoir under DC-offset AC electric fields. The effects of AC to DC field ratio and AC field frequency on the separation performance are both examined. We also develop a numerical model to understand and predict the observed cell motions in microfluidic reservoirs.

5:19PM L18.00009 Surface Design for Efficient Capturing of Rare Cells in Microfluidic Device , YALING LIU, Lehigh University, DAN DEPIETRO, ANTONY THOMAS, CHI-MON CHEN, SHU YANG, University of Pennsylvania — This work aims to design, fabricate, and characterize a micro-patterned surface that will be integrated into microfluidic devices to enhance particle and rare cell capture efficiency. Capture of ultralow concentration of circulating tumor cells in a blood sample is of vital importance for early diagnostics of cancer diseases. Despite the significant progress achieved in development of cell capture techniques, the enhancement in capture efficiency is still limited and often accompanied with drawbacks such as low throughput, low selectivity, pre-diluting requirement, and cell viability issues. The goal of this work is to design a biomimetic surface that could significantly enhance particle/cell capture efficacy through computational modeling, surface patterning, and microfluidic integration and testing. A PDMS surface with microscale ripples is functionalized with epithelial cell adhesion molecule (EpCAM) to capture prostate cancer PC3 cells. Our microfluid chip with micropatterns has shown significantly higher cell capture efficiency and selectivity compared to the chips with plane surface or classical herringbone-grooves.

5:32PM L18.00010 Cascade and staggered dielectrophoretic cell sorters , FANG YANG, XIAOMING YANG, HONG JIANG, GUIREN WANG, UNIVERSITY OF SOUTH CAROLINA TEAM, WJB DORN VETERANS AFFAIRS MEDICAL CENTER, COLUMBIA TEAM, GRACEFLOW TECHNOLOGY, IRMO, SC. TEAM — We report experimental results of successful separation of different cancer cells (breast and prostate) from colorectal cancer cells in Dielectrophoresis (DEP) in continuous operation. Conductivity influence on DEP spectrum for each cell type has been investigated. Under optimized condition, different cell type can be separated from each other. Enrichment factor and cell purity are measured to characterize the performance of the DEP chip. AC voltage and frequency effect on the separation is measured. In practical lab-on-a-chip application highly purified cell and high flow rate are required. In order to increase the purity of the isolated cells, cascade DEP sorter is developed. To increase flow rate, staggered DEP sorter is developed. It is found that compared with single DEP sorter, adding cascade DEP sorter can significantly increase r the purity of the target cell. With the staggered chip, the flow rate can be increased without compromising enrichment factor.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L19 Rayleigh Taylor Instability II 322

3:35PM L19.00001 One-dimensional-turbulence simulations of reactive Rayleigh-Taylor turbulence[1] , ESTEBAN GONZALEZ, ALAN KERSTEIN, Sandia National Laboratories, DAVID LIGNELL, Brigham Young University — We consider the problem of reactive Rayleigh-Taylor turbulence in the Boussinesq framework, and model combustion with a reaction-progress-variable method, and a KPP reaction. The interesting feature of this problem is that the interface (flame) between heavy/cold reactants and light/hot products moves against gravity. Such problem is challenging because of the delicate interplay between turbulence, buoyancy, and reactions, and the wide separation between large and small scales. One model that has the capabilities to deal with these challenges is the one-dimensional-turbulence (ODT) model. In this talk, we discuss ODT results for non-reactive and reactive Rayleigh-Taylor turbulence, and compare them with those from direct numerical simulations (DNS). Here, the key advantage of ODT over DNS is that it can be used to explore larger parameter spaces.

[1]This work was supported by the U.S. Department of Energy, Office of Basic Energy Sciences, Division of Chemical Sciences, Geosciences, and Biosciences.

3:48PM L19.00002 Late-time evolution of Rayleigh-Taylor instability in a domain of a finite size[1]

SNEZHANA ABARZHI, University of Chicago — For the first time a theoretical analysis was developed to systematically study the late-time evolution of Rayleigh-Taylor instability in a domain of a finite size. The nonlinear dynamics of fluids with similar and contrasting densities are considered for two-dimensional and three-dimensional flows driven by sustained or time-dependent acceleration. The flows are periodic in the plane normal to the direction of acceleration and have no external mass sources. Group theory analysis is applied to accurately account for the mode coupling. Asymptotic nonlinear solutions are found to describe the interface dynamics far from the boundaries and near the boundaries. The influence of the size of the domain on the diagnostic parameters of the flow is identified. In particular, it is shown that in a finite size the domain the flow is decelerating compared to spatially extended case. The theory outcomes for the numerical modeling and design of experiments on Rayleigh-Taylor instability are discussed.

[1]The work is supported by the US National Science Foundation and by the US Department of Energy.

4:01PM L19.00003 Reynolds–Averaged Navier–Stokes Modeling of Large Reynolds Number Mechanical and Scalar Rayleigh–Taylor Turbulent Mixing[1]

GREGORY BURTON, OLEG SCHILLING, Lawrence Livermore National Laboratory — A three- and four-equation, variable-density, incompressible Reynolds-averaged Navier–Stokes model incorporating mechanical and scalar turbulence is used to simulate Rayleigh–Taylor turbulent mixing with an Atwood number equal to one-half. Using both Reynolds number-dependent (optimal) and constant late-time model coefficients obtained by minimizing the L^2 norm between the model and the large Reynolds number 3072^3 Cabot–Cook direct numerical simulation data, the predicted mixing layer evolution is compared with the averaged DNS data in a posteriori tests. The terms in the transport equation budgets are compared in detail to their profiles across the mixing layer predicted by the DNS. The capability of the model to predict the degree of molecular mixing is also assessed.

[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.

4:14PM L19.00004 ABSTRACT WITHDRAWN —

4:27PM L19.00005 Turbulent Rayleigh-Taylor flow driven by time-varying accelerations

PRAVEEN RAMAPRABHU, University of North Carolina, Charlotte, ANDREW LAWRIE, LMFA, Ecole Centrale de Lyon, KARTHIK MUTHURAMAN, University of North Carolina, Charlotte, UNC-LMFA COLLABORATION — We report on numerical simulations of turbulent Rayleigh-Taylor flow subject to variable acceleration histories. The acceleration profiles were inspired by experiments and theoretical studies, and include an impulsive acceleration, accel-decel profiles, as well as a constant drive as the baseline case. The simulations were performed using the MOBILE software, a variable-density, incompressible fluid flow code. The advection algorithm employs a 3^{rd}-order, monotonicity-preserving upwind scheme, allowing the definition of sharp interfaces in the flow, while pressure convergence is accelerated by the use of a multi-grid scheme. The simulations are initialized with two classes of perturbations: narrow-band, short-wavelength modes and broadband with long-wavelength modes. The effect of initial amplitudes on the perturbations is investigated under the variable drive conditions. The acceleration profiles are capable of producing stages of "demixing," useful in validating turbulence models of RTI.

4:40PM L19.00006 Direct Numerical Simulations of Rayleigh-Taylor instability

DANIEL LIVESCU, TIE WEI, MARK PETERSEN, Los Alamos National Laboratory — The development of the Rayleigh-Taylor mixing layer is studied using data from an extensive new set of Direct Numerical Simulations (DNS). This includes a suite of simulations with grid size of $1024^2 \times 4608$ and Atwood number ranging from A=0.04 to 0.9, in order to examine small departures from the Boussinesq approximation as well as large Atwood number effects, and a high resolution simulation of grid size $4096^2 \times 4032$ and Atwood number of 0.75. After the layer width had developed substantially, additional branched simulations have been run under reversed and zero gravity conditions. The results presented address the role of the initial conditions on the mixing layer development and the discrepancy between the growth rates in various experiments and numerical simulations, as well as the changes in Rayleigh-Taylor turbulence properties at large density ratios.

4:53PM L19.00007 Simulations of Compressible Rayleigh-Taylor Instability Using the Adaptive Wavelet Collocation Method[1]

SCOTT J. RECKINGER, University of Colorado at Boulder, DANIEL LIVESCU, Los Alamos National Laboratory, OLEG V. VASILYEV, University of Colorado at Boulder — Numerical simulations of the single-mode compressible Rayleigh-Taylor instability are performed on an adaptive mesh using 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 with the solutions from linear stability theory, with two diffusively mixed, stratified fluids of differing molar masses as the background state. Of interest are the compressibility effects on the departure time from the linear growth, the onset of strong non-linear interactions, and the late-time behavior of the fluid structures. The late time bubble and 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.

5:06PM L19.00008 ABSTRACT WITHDRAWN —

5:19PM L19.00009 On Buoyancy and Shear Mixing

BETH PLACETTE, BHANESH AKULA, Texas A&M University, MALCOLM ANDREWS, Los Alamos National Laboratory, DEVESH RANJAN, Texas A&M University — Combined Rayleigh Taylor and Kelvin Helmholtz instabilities play a significant role in a number of phenomena, most importantly inertial confinement fusion. Should the relationship between initial conditions and mixing be determined, then, in principle, the level of mixing could be controlled through the setting of specific conditions. To investigate this proposition, a Kelvin Helmholtz Rayleigh Taylor experiment with a low Atwood number, buoyancy- and velocity-driven mixing width was investigated. The experiment was modeled using an implicit large eddy simulation code which uses a finite volume technique to solve the three dimensional incompressible Euler equations. The number of modes and the magnitude of the perturbations were set to investigate the rate of development as well as the maximum growth reached of the mixing width. Preliminary results show a promising overall agreement for the mixed region when the number of modes is increased.

5:32PM L19.00010 Direct Numerical Simulation of Tilted Rayleigh-Taylor Instability

TIE WEI, Los Alamos National Lab., DANIEL LIVESCU, Los Alamos National Lab — The tilted Rayleigh-Taylor instability, where the initial interface is not perpendicular to the driving acceleration, is investigated using Direct Numerical Simulations (DNS). In this configuration, the inclination of the initial interface results in a large-scale overturning motion in addition to the buoyancy driven instability. The DNS results are compared to the rocket-rig experiments of Smeeton and Youngs (AWE Report No. 35/87) at several Atwood numbers (A=0.267, 0.48, and 0.90). Since the initial conditions in these experiments are largely unknown, an extensive range of initial conditions have been explored to match the mixing layer growth between DNS and experiments. The evolution of the mixing layer was found to be strongly influenced, for the duration of the experiments, by the initial spectrum shape and peak location, as well as the perturbation amplitude. A set of initial conditions matching the experimental growth rates has been determined. Results are also presented on the interaction between shear and buoyancy, including the parameters influencing the overturning and mixing.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L20 Interfacial/Thin Film Instability V 323

3:35PM L20.00001 Linear Instability, Self-Similarity and Nonlinear Growth in Ultrathin Wetting Films Driven by Thermocapillary Flow , RYAN DENLINGER, SANDRA TROIAN, California Institute of Technology, 1200 E. California Blvd MC 128-95, Pasadena, CA — Nanoscale viscous films subject to very large thermocapillary stresses are susceptible to a linear instability resembling nanopillar arrays. The interstitial regions, which provide the fluid needed to grow these structures, are observed to undergo rapid depletion down to tens of nanometers in thickness. Previous analyses have neglected the role of van der Waals forces in these systems [1]. A linear stability analysis of ultrathin films in which the disjoining pressure is comparable to thermocapillary and capillary forces confirms that wetting van der Waals forces generate larger interpillar spacings. More importantly, however, growth beyond the linear regime is characterized by an ultra flat depletion zone. Using a combination of analytic work and finite element simulations, we have identified a self-similar regime in which finite-amplitude disturbances to this zone produce a significant non-linear response. The resulting waveform shape is indicative of a secondary growth process leading to nanopillar formation. We discuss the behavior of three distinct growth regimes characterizing initial pillar growth, depletion zone formation, and secondary pillar growth. [1] M. Dietzel and S. M. Troian, Phys. Rev. Lett. 103, 074501 (2009); J. Appl. Phys. 108, 074308 (2010)

3:48PM L20.00002 The onset of Marangoni convection for evaporating liquids with spherical interfaces and finite boundaries , BRENDAN MACDONALD, CHARLES WARD, University of Toronto — We examine the stability of evaporating liquids with spherical interfaces bounded at one value of the polar angle for all azimuthal angles. A linear stability analysis is performed and the results are used to explain why a system with water evaporating from a funnel constructed of PMMA undergoes stable quiescent evaporation but a system with a funnel constructed of stainless steel experiences a transition from a quiescent state to a state with Marangoni convection. We develop the expression for a new stability parameter that provides a quantitative prediction of the transition to Marangoni convection, and find the predictions to be consistent with experiments.

4:01PM L20.00003 Coherence Resonance Behavior in Thermocapillary Lithography , NAN LIU, SANDRA TROIAN, California Institute of Technology, 1200 E. California Blvd. MC 128-95, Pasadena, CA — Interest in alternative means of nanofilm lithographic patterning has focused attention on a number of thin film hydrodynamic instabilities which can spontaneously generate periodic arrays of 3D protrusions. The interface evolution equation, which results from a competition between stabilizing capillary forces and destabilizing external driving forces, are well described by a nonlinear, fourth order PDE rather sensitive to initial and boundary conditions. Even small levels of noise result in arrays prone to variations in array pitch and array height at levels currently unacceptable for commercial applications. In this talk, we focus on thin film patterning by thermocapillary forces. We demonstrate how an adjacent cooled template with a small sinusoidally roughened surface presented to the free surface of a molten nanofilm can be used to trigger very rapid and uniform array growth with a pitch even smaller than predicted by linear stability analysis. This behavior is reminiscent of coherence resonance phenomena in which a small amount of external noise can trigger resonant uniform growth. We quantify the behavior of waveforms generated in this way by a combination of weakly non-linear analysis and finite element simulations.

4:14PM L20.00004 Film falling on a porous substrate[1] , ARGHYA SAMANTA, CHRISTIAN RUYER-QUIL, Université Pierre et Marie Curie (UPMC), BENOÎT GOYEAU, Ecole Centrale Paris, FAST CNRS UMR 7608 TEAM, EM2C CNRS UPR 288 TEAM — Consider a two dimensional viscous incompressible liquid film falling on a saturated porous inclined plane. The interface between the liquid and porous medium is modeled using a one-domain approach for which the permeability and porosity varies continuously. A two-equation model is derived in terms of the flow rate $q(x,t)$ and total height $H(x,t)$ within the framework of boundary layer approximations using weighted residual techniques. Coefficients of the model are expressed in terms of combinations of the integrals of the base flow f and weight function w that are determined numerically to ensure consistency of the approach at first order in the film parameter. The influence of properties of the homogeneous porous substrate on the wave dynamics is investigated by constructing the nonlinear traveling wave solutions.

[1]Research partially supported by EU/FP7-ITN-MULTIFLOW.

4:27PM L20.00005 How does a soap film burst during generation?[1] , EMMANUELLE RIO, LAURIE SAULNIER, FREDERIC RESTAGNO, DOMINIQUE LANGEVIN, LPS, Universite Paris Sud — Foams are dispersions of bubbles in a liquid matrix in the presence of stabilizing surfactants. Even if foams are ubiquitous, the ability of a solution to create a certain foam quantity is still not fully understood. As a first step, we choose to work on a simplified system and studied the stability of a soap film during its generation. We have built an experiment, in which we determine simultaneously the velocity of a frame pulled out of a soapy solution and the entire shape of the liquid film. We found that the film is made of two parts: the bottom part is of uniform and stationary thickness, well described by the classical Frankel's law; in the top part, the film drains until a black film appears near the frame upper boundary frame, and then bursts. In this study, we characterize both part of the films and show that the Frankel law breaks down at high capillary number due to surfactants confinement. We also explain why films pulled at high velocity have a shorter lifetime than those pulled at low velocity.

[1]L. Saulnier is funded by CNES.

4:40PM L20.00006 Modeling of the stability of free-falling liquid curtain flow , FORTUNATO DE ROSA, Universita di Napoli Federico II, Dept. of Aerospace Engineering DIAS, Italy, GENNARO COPPOLA, Universita di Napoli Federico II, Dept. of Energetics and Applied Thermo-fluid-dynamics DETEC, Italy, LUIGI DE LUCA, Universita di Napoli Federico II, Dept. of Aerospace Engineering DIAS, Italy — The physical mechanisms leading to the disintegration of a gravitational (non parallel) two-dimensional plane liquid curtain (sheet), occurring at low fluid flow rates, are not yet fully known. The problem is reconsidered here through the development of an unsteady inviscid mathematical model where the dependent variables are expressed by means of polynomial expansions in terms of powers of the local lateral distance from the centerline position. Surface tension effects are included, and the ambient pressure field may be either applied or induced by the compliant free interface. The linearization around the base flow allows the separation of sinuous and varicose responses. The global linear stability of such a model is analyzed by inspecting both modal and non-modal amplifications of disturbances energy. An equation of energy budget is also derived, which is used to estimate the contribution of the various physical effects evaluated via direct numerical simulations of the governing system of equations.

4:53PM L20.00007 Layer formation in particle-laden free-films[1] , LUCIEN BRUSH, University of Washington, STEVEN ROPER, University of Glasgow — Solutions to a model of a particle-laden free-film that includes structural oscillatory forces in addition to van der Waals forces are presented. Examination of steady solutions to the equations reveals layer and bulge solutions. Fully nonlinear time-dependent numerical calculations reveal that at fixed concentration a uniform film evolves into a multi-layered film, the heights of which are given by the common tangent construction applied to the particle-film interaction free energy. If the interaction free energy curvature is negative there is no barrier to the formation of a layered film from the uniform film, whereas if the interaction free energy curvature is positive the uniform layer is metastable. This behavior is analogous to spinodal decomposition and nucleation and growth mechanisms observed in classical first order phase transformations. Phase diagrams for layer transitions are also presented and discussed.

[1]Support from NSF Grant No. CMMI-0827101 is acknowledged.

5:06PM L20.00008 Interfacial instabilities of reactive fingering , ANDONG HE, Institute for Computational and Experimental Research in Mathematics, Brown University, ANDREW BELMONTE, Department of Mathematics, Penn State University — We consider viscous flows in a Hele-Shaw cell in which two immiscible fluids chemically react and form a complex substance at the interface. The interface is modeled as an elastic membrane whose bending rigidity depends on the local curvature. A dispersion relation is derived using the energy variation method. Several types of instabilities are categorized and how various physical parameters affect the stability is investigated. Our model is able to explain the anomalous fingering instability from experimental observations reported by other authors.

5:19PM L20.00009 ABSTRACT WITHDRAWN —

5:32PM L20.00010 Thin film rupture on flat and structured surfaces with surface charge densities , CHRISTIAAN KETELAAR, VLADIMIR AJAEV, Southern Methodist University — We perform a linear and nonlinear stability analysis to determine the conditions at which a thin film of viscous liquid containing a small concentration of ions will rupture for different surface charge densities at the solid-liquid and gas-liquid interfaces. The rupture is driven by the combined action of the electrostatic component of the disjoining pressure and van der Waals forces. The evolution of the interface shape is described using the system of lubrication-type equations. By considering a small perturbation to the constant steady-state solution, we obtain the growth rate of the instability and find a wave number range where the perturbation decays. The nonlinear stability analysis is performed by solving the interface shape equation numerically for a range of parameters corresponding to different values of the initial film thickness, Debye length, and surface charge densities. We then discuss applications of the same mathematical framework to analyze film rupture on charged structured surfaces.

Monday, November 21, 2011 3:35PM - 5:32PM —
Session L21 Vortex Dynamics V 324-325

3:35PM L21.00001 Effects of Spanwise-Modulated Blowing on the Cylinder Near-Wake[1] , JAMES GREGORY, SAMIK BHATTACHARYA, The Ohio State University — Three-dimensional characteristics of the wake of a circular cylinder at Re=5×10^3 were controlled using spanwise-modulated forcing from dielectric barrier discharge plasma actuators. Three-dimensional blowing profiles were created by varying the shape of the buried electrode in the spanwise direction. A sinusoidal voltage of 10 kVpp at 5 kHz was applied between the electrodes to create the plasma. Wake surveys conducted with a rake of x-wires were used to understand the impact of the actuation on the time-resolved wake development. Simultaneous transverse profiles at multiple spanwise positions reveal that the actuation has a substantial impact on the spanwise variation of frequency and phase of the Kármán shedding process, as well as on the mean and fluctuating properties of the wake profile. Cross-correlations indicate a loss of coherence between adjacent spanwise locations, and the amplitude of the dominant vortex shedding frequency was substantially attenuated due to forcing. Further studies were carried out by modulating the actuation with harmonics and sub harmonics of Kármán shedding and the shear layer instability, with the aim of introducing small-scale structures that enhance break-up of the wake coherence.

[1]This work is supported by the Air Force Office of Scientific Research, Award Number FA9550-10-1-0490.

3:48PM L21.00002 A cantilevered flexible cylinder in cross-flow[1] , JESSICA SHANG, ALEXANDER SMITS, HOWARD STONE, Princeton University — Biological fluid-structure interactions of high aspect ratio bluff bodies are commonplace: flow around tall plants; flow through arrays of sensory vibrissae, antennae, and hairs. In this study, we seek insight to this class of problems by generalizing the flow configuration to uniform flow past a flexible cantilevered cylinder. Experiments were conducted for Re_D = 100-500. Cylinders deflected with the flow and demonstrated multimodal oscillations in both the streamwise and transverse directions. Oscillation frequencies were correlated with vortex shedding frequencies, but low oscillation frequencies (sub-1 Hz), which were not apparently vortex- induced, were also present. Two Re_D regimes were noted in which the vortex shedding frequency remained relatively constant with Re_D,while the two regimes were separated by an intermediate transition region. This feature results in an apparently linear relationship between St and Re_D in each regime. Hydrogen bubble visualization showed strong three-dimensionality in the wake, as well as a diversity of wake structures varying with Re_D.

[1]NSF-GRFP

4:01PM L21.00003 A Case Study of Horseshoe Vortex Dynamics at Laminar and Turbulent Juncture Flows , TAYFUN AYDIN, ADAM BLACKMORE, ALIS EKMEKCI, University of Toronto — To better understand the impacts of oncoming boundary-layer characteristics on the horseshoe vortex dynamics forming at a cylinder-wall junction, a case study is conducted in a free-surface water channel involving cylinder-channel floor and cylinder-end plate junctions. Endplates had two different leading-edge geometries, namely a sharp and a super-elliptical shape. The Reynolds number was kept at 10,000, based on the cylinder diameter. Laminar boundary layer forms along the channel floor and on the endplate with the super-elliptical leading edge. The plate with a sharp leading edge, however, involves flow separation at the leading edge, followed by a reattachment then an early transition to turbulence. On this plate, turbulent boundary layer is achieved. Flow visualization was performed on the plane of symmetry via PIV. Vortex trajectories and the velocity spectra depict periodic nature of the laminar horseshoe vortices and a variation of the spatial and temporal characteristics of horseshoe systems with the boundary layer thickness. Turbulent approach boundary layer led to disorganized and complex horseshoe vortex formation.

4:14PM L21.00004 Self-Excited Fluidic Energy Harvesters with Finite-Length Cylinders[1] , HUSEYIN DOGUS AKAYDIN, The City College of New York, CHLOE DUQUESNOIS, INSA de Lyon, France, NIELL ELVIN, YIANNIS ANDREOPOULOS, The City College of New York

— In this experimental work, we explore the possibility of using piezoelectric materials for harvesting electrical energy from fluid flow. Such harvesters may be used for powering small sensors and obviate the need for batteries and/or power lines. Piezoelectric harvesters behave as AC-coupled devices and need oscillatory motion to generate an electrical output. The harvester should be designed to be "self-excited," i.e. capable of initiating and sustaining the necessary oscillations in steady and uniform flows. The present configuration consists of a piezoelectric cantilever beam with a cylindrical tip body which promotes aeroelastic vibrations induced by vortex shedding. The harvester was tested in a wind tunnel and it produced 0.1 mW of electrical power at a flow speed of about 1.19 m/s. Using strain measurements and a distributed parameter model, the harvested electrical power was predicted, and a reasonable agreement is obtained with the measurements. The magnitude and frequency of the driving aerodynamic forces were also estimated. Results were comparable with literature data on flow past oscillating cylinders. Finally, the effect of using various shapes of tip body is presented.

[1]Sponsored by NSF Grant: CBET #1033117.

4:27PM L21.00005 Vortex Induced Vibration of an Inclined Cylinder in Flow , ANIL JAIN, YAHYA MODARRES-SADEGHI

— The Vortex Induced Vibration (VIV) of a cylinder inclined to the incoming flow is not studied as extensively as the normal-incidence case. For a rigid inclined cylinder, it is believed that the cylinder behaves similarly to a vertical one if the component of the flow velocity normal to the cylinder axis is considered. We investigate the extent to which this assumption is valid by conducting a series of experiments on a flexibly mounted cylinder placed inclined to the incoming flow with various angles of inclination in a Reynolds number range of 500 - 10,000. The cylinder, mounted on springs, is placed in the test section of a recirculating water tunnel, and for each angle of inclination, we increase the flow velocity gradually, and measure the displacement and frequency of the resulting vibrations. We examine how various angles of inclination result in various lock-in regions (large-amplitude vibrations) and we determine the critical angle of inclination beyond which the axial component of the flow velocity will have a non-negligible influence on the observed vibrations.

4:40PM L21.00006 Vortex dislocations in the wake of a circular cylinder , SERHIY YARUSEVYCH, CHRISTOPHER MORTON, University of Waterloo

— Cross flow over complex cylindrical geometries, such as cylinders with discontinuities in diameter, results in non-uniform wake vortex shedding with complex vortex interactions. In the present study, the development of vortex dislocations in the wake of a circular cylinder is investigated. A dual-step circular cylinder geometry is employed, which consists of a large-diameter cylinder placed at the midspan of a small-diameter cylinder. In a uniform flow, wake development depends primarily on the Reynolds number (Re_D), the ratio of the large cylinder diameter (D) to the small cylinder diameter (d), and the aspect ratio of the large cylinder (L/D). Experimental investigations are performed for Re_D=1080, D/d=2, and $0.2 \leq L/D \leq 2.0$ in a water flume facility utilizing flow visualization, laser Doppler velocimetry, and particle image velocimetry. Also, direct numerical simulations are performed for a laminar vortex shedding regime, where, based on previous studies, interactions between large-scale vortical structures are expected to be similar to those observed in turbulent wakes. The results show that the large cylinder induces vortex dislocations that are manifested by half-loop vortex connections forming between consecutive small cylinder vortices. The remaining small cylinder vortices form vortex connections across the wake of the large cylinder. The dislocations occur at a distinct frequency, which decreases with decreasing L/D.

4:53PM L21.00007 Nonlinear restoring forces in vortex-induced vibration , A.W. MACKOWSKI, C.H.K. WILLIAMSON, Cornell University

— When studying vortex-induced vibration of a rigid circular cylinder, almost all experimental and computational studies involve the cylinder being supported by linear springs. However, there are cases in which we may be interested in the VIV response of a cylinder supported by nonlinear springs. A system with nonlinearities in the restoring force has the potential to increase the amplitude response envelope, critical to the success of aero-vibrating energy harvesters. On the other hand, designing nonlinear restoring forces to decrease the amplitude response may lead to structures more able to withstand flow-induced vibration. In addition, adding nonlinear terms to the restoring force on a rigid cylinder might be used to simulate higher-order dynamics of long, elastic marine cables. To experimentally observe the effects of nonlinear springs on flow-induced vibration, we apply a novel approach that lets us parametrically control the nature of the springs and the strength of the nonlinearities. The technique, called Cyber-Physical Fluid Dynamics, uses a force-feedback control system to simulate arbitrary forces on a submerged body [the details of this system were shown in the APS presentation of Mackowski & Williamson (2010)]. We present results using this technique to explore the amplitude response of a circular cylinder in a crossflow.

5:06PM L21.00008 The flow around the node to anti-node transition of an oscillating flexible cylinder[1] , FRANCISCO HUERA-HUARTE[2], ZAFAR BANGASH[3], Universitat Rovira i Virgili

— The vortex dynamics in the wake of cylinders is a topic of interest, for its implications in engineering since the forties. Most of the work has been done with stationary cylinders or with oscillating rigid cylinders when they are elastically mounted. If the cylinders are flexible, not only the structural dynamics become more complex, but the fluid dynamics around the cylinder are much more complicated. We study the flow around a flexible cylinder when oscillating in its first or second structural mode. An experimental set-up allows producing forced oscillations on the model, in a way that mode number, oscillating frequency and amplitude can be controlled. Experiments were conducted in a motorised towing tank, for a wide range of towing speeds, frequencies and amplitudes. Digital Particle Image Velocimetry (DPIV) was used to quantify the flow and to investigate the flow structures and vortex modes around the cylinder. The interest was given to the extreme amplitude regions and to the node to anti-node transitional regions.

[1]Spanish Ministry of Science (MICINN) grant DPI2009-07104 is acknowledged.

[2]Associate Prof., Department of Mechanical Engineering

[3]PhD student, Department of Mechanical Engineering

5:19PM L21.00009 The Effect of Shape on the Wake of Low-Aspect-Ratio Wall-Mounted Obstacles[1] , SEYED HAJIMIRZAIE, JAMES BUCHHOLZ, University of Iowa

— Wall-mounted bodies in boundary layer flows are ubiquitous in nature and engineering applications. We evaluate the role of shape on the wakes around three different low-aspect-ratio wall-mounted obstacles in shallow boundary-layer flow: semi-ellipsoids with the major axis of the base ellipse aligned in the transverse and streamwise directions, and a sphere. Despite their geometric simplicity, the obstacles create extremely complex, highly three-dimensional and unsteady flow fields for which the transport mechanisms of momentum and scalars are still not well-understood. All three obstacles have the same height and the aspect ratios considered are 0.67, 0.89 and 1, respectively. DPIV was used to interrogate the flow. Streamwise structures observed in the mean wake include tip, base, and horseshoe vortex pairs, which vary significantly in strength with changes in obstacle geometry. Significant variation in the strength of these structures with streamwise location suggests a complex connectivity with the mean spanwise arch structure in the near wake. The three-dimensional topology of the mean wake will be discussed.

[1]This work is supported by NSF grant number CBET-1033732.

Monday, November 21, 2011 3:35PM - 5:58PM –
Session L23 Low Re Number Aerodynamics 326

3:35PM L23.00001 Spanwise drag variation on low Re wings – revisited , SHANLING YANG, GEOFFREY SPEDDING, University of Southern California — Aerodynamic performance measurement and prediction of airfoils and wings at chord Reynolds numbers below 10^5 is both difficult and increasingly important in application to small-scale aircraft. Not only are the aerodynamics strongly affected by the dynamics of the unstable laminar boundary layer but the flow is decreasingly likely to be two-dimensional as Re decreases. The spanwise variation of the flow along a two-dimensional geometry is often held to be responsible for the large variations in measured profile drag coefficient. Here we measure local two-dimensional drag coefficients along a finite wing using non-intrusive PIV methods. Variations in $C_d(y)$ can be related to local flow variations on the wing itself. Integrated values can be compared with force balance data, and the proper description of drag components at low Re will be discussed.

3:48PM L23.00002 Exploration of the Relationship Between Wake Vortex Parameters and Thrust Force on Oscillating Airfoils Using a Vortex Array Model[1] , AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — Recently, we demonstrated the ability of a simple model, based on an array of finite-core Gaussian vortices, to accurately reproduce the unsteady velocity field in the wake of, and drag/thrust force acting on harmonically/non-harmonically pitching airfoils. In the present work, this model is employed to explore how the thrust force varies with wake vortex parameters; i.e. circulation, core radius and streamwise/cross-flow spacing of the vortices. Insight from this investigation will be helpful to draw links between trailing-edge flexibility and the detailed process of generation of wake vortices. Such links may have the potential for providing a path towards a rational, yet efficient, approach for tailoring trailing-edge flexibility to obtain desirable force characteristics for flapping-wings Micro Air Vehicles.

[1]Supported by AFOSR Grant No. FA9550-10-1-0342.

4:01PM L23.00003 Vortex arrangement in the wake of rigid and flexible rapidly pitching airfoils at low Reynolds number[1] , BRUNO MONNIER, AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — An experimental investigation of the wake of an airfoil undergoing rapid pitch oscillation is conducted in a water tunnel at a chord Reynolds number of about 2000. Flow visualization is utilized to characterize the vortical patterns in the wake of the airfoil, which is constructed from a NACA 0036 profile fitted with an extended trailing edge with controllable flexibility. The spatial configuration of the vortices is extracted in terms of streamwise and cross-flow spacing over a range of pitching frequencies and amplitudes. We discuss how different levels of flexibility alter the vortex spacing parameters and the conditions under which the traditional Karman vortex pattern, corresponding to a wake profile, changes to the reverse Karman pattern associated with a jet profile.

[1]Supported by AFOSR Grant No. FA9550-10-1-0342.

4:14PM L23.00004 Effects of fluid behavior around low aspect ratio, low Reynolds number wings on aerodynamic stability , MATTHEW SHIELDS, KAMRAN MOHSENI, University of Florida — The innovation of micro aerial vehicles (MAVs) has brought to attention the unique flow regime associated with low aspect ratio (LAR), low Reynolds number fliers. The dominant effects of developing tip vortices and leading edge vortices create a fundamentally different flow regime than that of conventional aircraft. An improved knowledge of low aspect ratio, low Reynolds number aerodynamics can be greatly beneficial for future MAV design. A little investigated but vital aspect of LAR aerodynamics is the behavior of the fluid as the wing yaws. Flow visualization experiments undertaken in the group for the canonical case of varying AR flat plates indicate that the propagation of the tip vortex keeps the flow attached over the upstream portion of the wing, while the downstream vortex is convected away from the wing. This induces asymmetric, destabilizing loading on the wing which has been observed to adversely affect MAV flight. In addition, experimental load measurements indicate significant nonlinearities in forces and moments which can be attributed to the development and propagation of these vortical structures. A non-dimensional analysis of the rigid body equations of motion indicates that these nonlinearities create dependencies which dramatically change the conventional linearization process. These flow phenomena are investigated with intent to apply to future MAV design.

4:27PM L23.00005 The Effects of AR on Membrane Wing Performance in Low Re Flight[1] , ALEX JORDAN, Purdue University, JAMES HUBNER, University of Alabama — There is increased interest in the design of micro air vehicles (MAVs) due to their military reconnaissance and surveying capabilities. Research has shown that the use of membrane wings in low Reynolds number flight results in performance characteristics that, when compared to rigid wing counterparts of similar geometry, are beneficial. An experimental study was performed to determine if the benefits of membrane wings change when AR is decreased. The membrane wings used silicon rubber affixed to aluminum frames of repeated cell geometry. The wings tested employed 1, 3, 5 and 9 cells and had ARs of 0.9, 2.6, 4.1, and 4.33 respectively. Measurements of lift and drag at a Reynolds number of 49,000 were acquired over a range of angles of attack. Vibration frequencies of the membranes were obtained via high-speed imagery. Comparisons of lift and drag data for the flat plates and membrane wings showed that the membrane wings with ARs of 0.9 and 2.6 did not show the same performance benefits as the higher AR membrane wings.

[1]Funded by NSF REU Site #1062611.

4:40PM L23.00006 Volumetric Three-Component Measurements of the Flowfield around Periodically Cambered Plates[1] , REDHA WAHIDI, ZHENG ZHANG, JAMES HUBNER, AMY LANG, The University of Alabama — The small size and low speed of the micro aerial vehicles (MAVs) place them in the low Reynolds number (Re) regime. The performance of conventional airfoils severely deteriorates at low Reynolds numbers. Therefore, unconventional wing designs are necessary to meet the operational requirements of the MAVs. Periodically cambered plates with an aspect ratio of five (AR = 5) were constructed for flowfield and performance measurements. Three-dimensional three-component (3D3C) flowfield measurements were carried out at a Reynolds number of 28,000 at different angles of attack. The locations of separation, transition and reattachment are estimated based on the mean velocities and Reynolds shear and normal stresses in the streamwise, spanwise and wall-normal directions. The effects of the tip vortices on the three-dimensionality of the reattachment line are resolved by the 3C3C measurement technique. Additionally, the effects of the cell size on the separation and reattachment locations are investigated. The results also include details about the vorticity about the X, Y and Z axes.

[1]National Science Foundation under Grant No. 0958668.

4:53PM L23.00007 Dynamics of Spanwise Vorticity on a Rotating Flat Plate , CRAIG WOJCIK, JAMES BUCHHOLZ, University of Iowa — Leading-edge vortex (LEV) structures were examined using phase-locking digital particle image velocimetry for two rotating flat plates with aspect ratios of 2 and 4. The plates were accelerated uniformly for approximately 45° to a constant rotational speed. Reynolds numbers of 4,000, 8,000, and 16,000 were investigated. The flow field was measured in chordwise planes at two spanwise positions, while varying angle of attack and azimuthal position of the plate. The results show a concentrated, stationary vortex structure at the leading-edge of the plate. The strength of the leading edge vortex (LEV) was found to vary with azimuthal position, Reynolds number, and angle of attack. The time variation in LEV strength is studied using a control volume analysis taking into account in-plane and out-of-plane vorticity fluxes as well as interactions with opposite-sign vorticity generated beneath the LEV on the plate surface.

5:06PM L23.00008 Vortex Shedding vs. Thrust Production for Free-to-Pivot Flat Plates , MICHAEL OL, KENNETH GRANLUND, US AIr Force Research Lab — As an abstraction of flapping-wings, we consider flat plates in rectilinear motion, with the leading edge undergoing periodic oscillation, and the plate left free to pivot about its leading edge, between incidence angle limits of $\pm 45^\circ$. Measurements include thrust production and resistive force, with leading edge and trailing edge vortices visualized by dye injection, conducted in a water tunnel operated here as a towing tank. Imposed acceleration of the plate's leading edge produces a rotational motion followed by a translational phase at constant incidence angle, and a reverse rotation at the semi-stroke extremum. Varying aspect ratio from 3.4 to nominally 2D, neither thrust nor resistive force evince an aspect ratio dependency. Reynolds number does not effect flow development or force production across $5000 < Re < 25000$. Investigating the conjecture that imposed acceleration stabilizes the leading edge vortex, we find no difference across a range of acceleration profiles. The dominant parameter affecting thrust production is the plate stroke to chord ratio, with values of $\sim$6 and above being most favorable. Further, as a simplification of aeroelastic effects conducted with otherwise rigid plates, we consider a plate sliced spanwise and thus forming two hinged plates. This produces both lower thrust and resistive force than in the single-plate case, resulting in no improvement in hovering figure of merit, which amongst all cases peaks at ~ 0.3.

5:19PM L23.00009 Fluid Structure Interactions on an Electro-Elastomer Membrane Wing , LAWRENCE UKEILEY, ADAM HART, University of Florida, MICHAEL HAYS, WILLIAM OATES, Florida State University, BENJAMIN DICKINSON, Air Force Research Laboratory — Wing flexibility is an important aspect of many natural flyers that has been demonstrated to have several specific aerodynamic benefits. An engineering abstraction of flexibility in wings is to use an extensible membrane stretched over a rigid frame. A key aspect of a membrane wing's performance is its tension as that will dictate how far it can stretch and its natural vibrational frequency. In this work we use a dielectric elastomer membrane, whose ability to stretch is a function of a voltage applied across it. The membrane is adhered to an elliptical planform wing which was placed in the freestream of an open jet wind tunnel. Measurements of the aerodynamic performance, membrane shape and flow field over the wing have been acquired as a function of angle of attack and voltage across the elastomer independently as well as synchronously. These measurements demonstrate how the membrane's characteristics alter the flow over it and translate to the generation of aerodynamic forces. These characteristics are studied in terms of both the static deflection increasing the wings camber as well as the time dependent excitation through the membrane's vibrations.

5:32PM L23.00010 Fluid and Structure Dynamics of Flow over a Membrane Wing , AMORY TIMPE, LAWRENCE UKEILEY, University of Florida — The coupled effect of flow induced membrane deformations and their return influence on the flow are investigated. Multi-cell wings are made by adhering heated, thin Silicone membranes to 2.7 percent thick, rectangular aluminum frames with rigid leading edge and battens. Time-resolved flow and structure deformations are measured by synchronized acquisition of high-speed, two-component Particle Image Velocimetry (PIV) and stereoscopic Digital Image Correlation (DIC). Mean and instantaneous effects are studied at a chord based Reynolds number of 48,000, while metrics are compared for membrane cell size of aspect ratio 1 and 0.5. Two PIV measurement fields of view are employed to study flow over the membrane surface as well as that of the near wake. Power spectral density and correlation techniques will be utilized, along with analysis of membrane mean deformation and rms fluctuation behavior to better understand the fluid-structure interactions. The effects of membrane behavior on flow separation, shear layer size and location, along with vorticity will be analyzed in comparison to flow over a similar geometry flat plate.

5:45PM L23.00011 Falling Maple Seed: The mechanism for its spin , Z. JANE WANG, Cornell University, KAPIL VARSHNEY, SONG CHANG, Cornell University — A maple seed falls in a characteristic downward spiral. One explanation of its spinning motion is the auto-rotational effect. As the seed falls, the aerodynamic torque on the feather will cause it to spin like a wind-mill. However, the film of a falling seed shows that its spinning motion precedes the steady descent. And surprisingly, the seed with little feather can also spin. We experimentally quantify the kinematics of falling seeds with different shapes. The analysis of the coupling between the rigid body dynamics and the aerodynamics offers a new explanation for the cause of the spin.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L24 Biofluids: General I 327

3:35PM L24.00001 Meshfree Computations for Flow Pumping in a Microchannel Inspired by Insect Physiological Systems , YASSER ABOELKASSEM, ANNE STAPLES, Virginia Tech — The present study is inspired by pumping mechanisms observed in physiological systems in insects that use multiple contractions to transport fluid. A meshfree computational method is used to solve for the 2D viscous flow in a microchannel at low Reynolds number. The channel is assumed to have a large aspect ratio and localized multiple contractions from the upper wall. These contractions are allowed to move with or without time (phase) lags with respect to each other. The flow development and structures induced by these wall contractions are obtained at various snapshots in the collapse cycle. The effect of the contraction amplitudes and time-lags between its individual prescribed motion protocols on the flow variables and on the time-averaged net flow over a complete cycle of wall motions is studied. The meshfree computational approach presented here is based on the method of fundamental solutions (MFS) which is considered to be an efficient numerical technique for solving elliptic boundary value problems (BVP) such as the Stokes equations. This class of numerical methods uses a set of singularized force elements (Stokeslets) which are distributed according to the method of collocations with unknown strengths. The Stokeslets' strengths are then calculated by imposing the appropriate boundary conditions and solving the resulting system of equations. The flow motions induced by these point forces are then computed by the principle of superposition.

3:48PM L24.00002 Butterfly proboscis: natural combination of a drinking straw with a nanosponge[1] , KOSTYA KORNEV, DARIA MONAENKOVA, PETER ADLER, Clemson University, WAH-KEAT LEE, Argonne National Lab, MATTHEW LEHNERT, TARAS ANDRUKH, CHARLES BEARD, BINYAMIN RUBIN, ALEXANDER TOKAREV, Clemson University — The ability of Lepidoptera, or butterflies and moths, to drink liquids from rotting fruit and wet soil, as well as nectar from floral tubes, raises the question of whether the conventional view of the proboscis as a drinking straw can account for the withdrawal of fluids from porous substrates or of films and droplets from floral tubes. We discovered that the proboscis promotes capillary pull of liquids from diverse sources due to a hierarchical pore structure spanning nano- and microscales. X-ray phase-contrast imaging reveals that Plateau instability causes liquid bridges to form in the food canal, which are transported to the gut by the muscular sucking pump in the head. The dual functionality of the proboscis represents a key innovation for exploiting a vast range of nutritional sources. A transformative two-step model of capillary intake and suctioning can be applied not only to butterflies and moths but also potentially to vast numbers of other insects such as bees and flies.

[1]NSF EFRI - 0937985

4:01PM L24.00003 Bubbles of Metamorphosis[1] , MANU PRAKASH, Stanford University — Metamorphosis presents a puzzling challenge where, triggered by a signal, an organism abruptly transforms its entire shape and form. Here I describe the role of physical fluid dynamic processes during pupal metamorphosis in flies. During early stages of pupation of third instar larvae into adult flies, a physical gas bubble nucleates at a precise temporal and spatial location, as part of the normal developmental program in Diptera. Although its existence has been known for the last 100 years, the origin and control of this "cavitation" event has remained completely mysterious. Where does the driving negative pressure for bubble nucleation come from? How is the location of the bubble nucleation site encoded in the pupae? How do molecular processes control such a physical event? What is the role of this bubble during development? Via developing in-vivo imaging techniques, direct bio-physical measurements in live insect pupal structures and physical modeling, here I elucidate the physical mechanism for appearance and disappearance of this bubble and predict the site of nucleation and its exact timing. This new physical insight into the process of metamorphosis also allows us to understand the inherent design of pupal shell architectures in various species of insects.

[1]Milton Award, Harvard Society of Fellows; Terman Fellowship, Stanford

4:14PM L24.00004 Contraction driven flow in the extended vein networks of *Physarum polycephalum* , KAREN ALIM, GABRIEL AMSELEM, FRANCOIS PEAUDECERF, School of Engineering and Applied Sciences, Harvard University, ANNE PRINGLE, Department of Organismic and Evolutionary Biology, Harvard University, MICHAEL P. BRENNER, School of Engineering and Applied Sciences, Harvard University — The true slime mold *Physarum polycephalum* is a basal organism that forms an extended network of veins to forage for food. *P. polycephalum* is renown for its adaptive changes of vein structure and morphology in response to food sources. These rearrangements presumably occur to establish an efficient transport and mixing of resources throughout the networks thus presenting a prototype to design transport networks under the constraints of laminar flow. The physical flows of cytoplasmic fluid enclosed by the veins exhibit an oscillatory flow termed "shuttle streaming." The flow exceed by far the volume required for growth at the margins suggesting that the additional energy cost for generating the flow is spent for efficient and/or targeted redistribution of resources. We show that the viscous shuttle flow is driven by the radial contractions of the veins that accompany the streaming. We present a model for the fluid flow and resource dispersion arising due to radial contractions. The transport and mixing properties of the flow are discussed.

4:27PM L24.00005 Fluid Flow of Vitrectomy , POORIA SHARIF-KASHANI, TINGTING JUAN, JEAN-PIERRE HUBSCHMAN, JEFF D. ELDREDGE, H. PIROUZ KAVEHPOUR, University of California, Los Angeles — Vitrectomy is a microsurgical technique to remove the vitreous gel from the vitreous cavity. Due to the viscoelastic nature of the vitreous gel, its complex fluidic behavior during vitrectomy affects the outcome of the procedure. Therefore, the knowledge of such behavior is essential for better designing the vitrectomy devices, such as vitreous cutters, and tuning the system settings such as port and shaft diameters, infusion, vacuum, and cutting rate. We studied the viscoelastic properties of porcine vitreous humor using a stressed-control shear rheometer and obtained its relaxation time, retardation time, and shear-zero viscosity. We performed a computational study of the flow in a vitreous cutter using the viscoelastic parameters obtained from the rheology experiments. We found significant differences between the modeled vitreous gel and a Newtonian surrogate fluid in the flow behavior and performance of the vitreous cutter. Our results will help in understanding of the vitreous behavior during vitrectomy and providing guidelines for new vitreous cutter design.

4:40PM L24.00006 Leonardo's branching rule in trees: How self-similar structures resist wind[1] , CHRISTOPHE ELOY, UC San Diego — In his notebooks, Leonardo da Vinci observed that "*all the branches of a tree at every stage of its height when put together are equal in thickness to the trunk*," which means that the total cross-sectional area of branches is conserved across branching nodes. The usual explanation for this rule involves vascular transport of sap, but this argument is questionable because the portion of wood devoted to transport varies across species and can be as low as 5%. It is proposed here that Leonardo's rule is a consequence of the tree skeleton having a self-similar structure and the branch diameters being adjusted to resist wind-induced loads. To address this problem, a continuous model is first considered by neglecting the geometrical details of branching and wind incident angles. The robustness of this analytical model is then assessed with numerical simulations on tree skeletons generated with a simple branching rule producing self-similar structures.

[1]This study was supported by the European Union through the fellowship PIOF-GA-2009-252542.

4:53PM L24.00007 Aquatic vegetation in flow: To buoy or not to buoy , MITUL LUHAR, HEIDI NEPF, Massachusetts Institute of Technology — Previous studies show that the flexural stiffness and buoyancy of many species of aquatic vegetation change in response to hydrodynamic conditions. We present a theoretical and experimental study that describes the flow-induced reconfiguration of aquatic vegetation across the natural range of vegetation buoyancy and stiffness. We show how posture and drag depend on two dimensionless parameters that represent the relative magnitudes of the hydrodynamic forcing, and the restoring forces due to stiffness and buoyancy. Reconfiguration leads to a transition away from the classical quadratic drag law. We present scaling laws that describe the relationship between drag and velocity for both stiffness- and buoyancy-dominated reconfiguration. Our results may explain the morphological plasticity observed for aquatic vegetation.

5:06PM L24.00008 Modeling tumor growth in a complex evolving confinement using a diffuse domain approach[1] , YAO-LI CHUANG, Pathology, Univ of New Mexico, JOHN LOWENGRUB, YING CHEN, XIANGRONG LI, Mathematics, Univ of California, Irvine, HERMANN FRIEBOES, Bioengineering, Univ of Louisville, VITTORIO CRISTINI, Pathology, Chemical Engineering, Univ of New Mexico — Understanding the spatiotemporal evolution of tumor growth represents an essential step towards engineering effective treatment for cancer patients. At the macroscopic scale, various biophysical models describing tumors as continuum fluids have been constructed, particularly on a Cartesian grid, where efficient numerical schemes are available to analyze the model for general tumor behaviors in a relatively unconfined space. For practical problems, however, tumors are often found in a confined sub-domain, which can even be dilated and distorted by the growing tumor within. To study such tumors, we adopt a novel diffuse domain approach that enables us to adapt a model to an evolving sub-domain and formulate the modified problem on a Cartesian grid to utilize existing numerical schemes. To demonstrate this approach, we adapt a diffuse-interface model presented in Wise et al. [2008, Three-dimensional multispecies nonlinear tumor growth - I Model and numerical method, J. Theor. Biol. 253, 524-543] to simulate lymphoma growth in a lymph node structure.

[1]Supported by NIH-PSOC grant 1U54CA143907-01.

5:19PM L24.00009 The mechanics of pollination by wind: is anemophily aeroelastically optimized for reproduction? , DAVID TIMERMAN, Department of Biology, Concordia University, Montreal, QC, DAVID F. GREENE, Department of Geography, Concordia University, Montreal, QC, JOSEF D. ACKERMAN, Department of Integrative Biology, University of Guelph, Guelph, ON, JAVIER URZAY, Center for Turbulence Research, Stanford University, Stanford, CA — Approximately 10 percent of plant species rely on wind for pollination (anemophily). These include many taxa of economic importance: e.g. cultigens such as wheat and maize; species like grasses and ragweed that trigger allergies; and the conifers, our most important species for the forest industry in the mid- latitudes. It has often been assumed that anemophily is an inefficient mechanism compared to animal pollination (zoophily), but very little is known about the forces and micromechanics that deliver pollen grains into wind streams. Here we ask a fundamental question: is anemophily optimized for pollen shedding? In this talk, we focus on an as-yet rudimentary theory of turbulence- initiated pollen shed that models the pollen-bearing stamen as an aeroelastic oscillator. Ongoing experiments with anemophilous and zoophilous flowers excited by shakers are analyzed to extract values of damping ratios, adhesion forces and flexural rigidities. Finally, the anatomical differences between anemophilous and zoophilous species are evaluated using a dimensionless number that measures the ratio of adhesion to aeroleastic forces.

5:32PM L24.00010 Universality of osmotically driven sap-flow in plants , TOMAS BOHR, KÅRE HARTVIG JENSEN, KIRSTINE BERG SØRENSEN, SØREN MØRCH FRIIS, Center for Fluid Dynamics, Department of Physics, Technical University of Denmark, JOHANNES LIESCHE, ALEXANDER SCHULZ, Department of Plant Biology and Biotechnology, University of Copenhagen — Since Ernst Münch in the 1920s proposed that sugar transport in the phloem vascular system of plants is driven by passive osmotic pressure gradients, it has been strongly debated whether this hypothesis can account even for long distance translocation. Recently, it was shown that theoretical optimization of the Münch mechanism leads to surprisingly simple predictions for the dimensions of the phloem sieve elements in relation to those of the plants [Jensen et. al., J. Roy. Soc. Interface **8**, pp. 1155–1165 (2011)]. We show that the theoretical results are very insensitive to the details of the sugar-loading (in leaves) and unloading (in shoots or roots) and can even be obtained from a simple coupled resistor model. We have compiled anatomical data for a wide group of plants and find good agreement with theory, even for conifer trees, in which the sugar translocation is substantially slower than hardwood trees.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L25 Particle-laden Flows IV 328

3:35PM L25.00001 Simulated tornado debris tracks: implications for inferring corner flow structure[1] , MICHAEL ZIMMERMAN, NASA Goddard Space Flight Center, DAVID LEWELLEN, West Virginia University — A large collection of three-dimensional large eddy simulations of tornadoes with fine debris have been recently been performed as part of a longstanding effort at West Virginia University to understand tornado corner flow structure and dynamics. Debris removal and deposition is accounted for at the surface, in effect simulating formation of tornado surface marks. Physical origins and properties of the most prominent marks will be presented, and the possibility of inferring tornado corner flow structure from real marks in the field will be discussed.

[1]This material is based upon work supported by the National Science Foundation under Grants No. 0635681 and AGS-1013154.

3:48PM L25.00002 Numerical simulation of a bidisperse turbidity current interacting with a Gaussian bump , MOHAMAD M. NASR-AZADANI, ECKART MEIBURG, UC Santa Barbara — We study a particle-laden lock-exchange current interacting with a Gaussian bump by means of DNS simulations. Our software package TURBINS employs an immersed boundary implementation of the Boussinesq Navier-Stokes equations for the fluid motion, coupled to transport equations for the particle concentration fields. The suspension includes two particle sizes with a settling velocity ratio of 10. As the current travels over the bottom topography, we record instantaneous deposit profiles and wall shear stress contours. As the current impinges on the obstacle, it becomes strongly three-dimensional. Comparison of the final deposit profiles near the Gaussian bump against the case of a flat surface shows a smaller influence of the topography on the fine particles than on the coarse ones. Due to lateral deflection, deposition generally decreases near the bump, while increasing away from it. Some distance downstream of the obstacle, the deposit profiles lose their memory of the bump and become nearly uniform again. Instantaneous wall shear stress profiles are employed in order to estimate the critical conditions at which bedload transport and/or particle resuspension can occur in various regions.

4:01PM L25.00003 Modelling Submarine Turbidity Currents , ALEXANDER GOATER, ANDREW J. HOGG, Centre for Environmental and Geophysical Flows, University of Bristol — When a large scale pyroclastic flow enters the ocean it leads to the spreading of sediment across the deep ocean through particle laden flows known as turbidity currents. Turbidity currents are driven by gravitational forces associated with a density difference caused by the presence of suspended particles. This generates a flow which transports the suspended particles, but which progressively slows as they sediment to the underlying boundary. We adopt a shallow layer model in which vertical accelerations are neglected and employ a three equation system that expresses the conservation of fluid and particulate mass and formulates a balance of momentum for a current flowing down an incline. Importantly we include the effects of entrainment of surrounding fluid into the flow. Solutions are constructed using numerical means and they reveal the strong dependence of run out length on the rate of entrainment. Further, the prediction of the distribution of the deposit from the flow compares favourably with field data from the July 2003 event from Soufrière Hills volcano, Montserrat.

4:14PM L25.00004 Convective instability in sedimentation , XIAO YU, TIAN-JIAN HSU, University of Delaware — We investigate the convective sedimentation in a stably stratified saltwater using linear stability analysis and 3D direct numerical simulation. We consider sediment particle with grain diameter in the range of 1.5 to 60 μm. Equilibrium Eulerian approach and dilute flow assumption are adopted to simplify the governing equations of the two-phase system (Balachandar and Eaton 2009, Annul Rev. Fluid Mech.). A semi-empirical closure of particle diffusivity due to long-range interaction is adopted (Segre et al. 2007, Phys. Rev. Lett.). For a fixed salt diffusivity, the particle phase can act as either slow or fast diffusing agent in a double-diffusive system depending on the particle diameter. Additionally, the settling-driven mechanism can also trigger instability. Linear stability analysis is carried out as the guideline for 3D numerical simulation. Simulation results indicate different finger patterns for different particle settling velocity and sediment concentration. For fine particle, where the double-diffusive mechanism plays an important role, the instability is enhanced by the settling. The finger size is on centimeter scale and the finger pattern is more nonlinear and asymmetric. For large particle, the interfacial instability appears long after the particles pass the density interface induced by salt where Rayleigh-Taylor instability takes place and finger pattern is more symmetric. Fully nonlinear analysis with 3D direct numerical simulations will be presented in the meeting.

4:27PM L25.00005 Particle jet formation during explosive dispersal of solid particles , DAVID FROST, YANN GREGOIRE, SAM GOROSHIN, McGill University, ROBERT RIPLEY, Martec Ltd., FAN ZHANG, DRDC Suffield — Previous experimental studies have shown that when a layer of solid particles is explosively dispersed, the particles often develop a non-uniform spatial distribution. The instabilities within the particle bed and at the particle layer interface likely form on the timescale of the shock propagation through the particles. The mesoscale perturbations are manifested at later times in experiments by the formation of coherent clusters of particles or jet-like particle structures, which are aerodynamically stable. Experiments have been carried out in spherical and cylindrical geometry to investigate the influence of particle diameter and density and the ratio of particle to high explosive mass on the relative tendency for instabilities to develop in the expanding particle cloud. The number of particle jets that form tends to scale with a particle compaction Reynolds number corresponding to the ratio of inertial to frictional forces of the particle system. Below a critical Reynolds number, the expanding particle cloud remains stable.

4:40PM L25.00006 Shear stress and particle removal measurements of a turbulent air jet impinging normally upon a surface , RYAN YOUNG, MICHAEL HARGATHER, GARY SETTLES, Gas Dynamics Lab, Penn State University — When a jet of air impinges normally upon a surface, it imposes a shear stress parallel to the wall in all directions from the impingement point. Particle removal from that surface is assumed to be mainly due to that imposed shear stress. But that shear stress has been difficult to measure and has, in the past, been inferred from particle removal rates. Here we make a basic measurement of mean shear stress imposed upon a planar wall by a normally-impinging turbulent air jet using the technique of oil-film interferometry. The resulting shear-stress distribution is then compared with the removal rates of latex microspheres from a planar glass surface as a function of the distance from jet impingement normalized by the height of the nozzle above the surface. The particle removal experiments are carried out with sparse (few particle collisions) particle distributions. These experiments show that the efficiency of particle removal is directly but not linearly related to the imposed shear stress. A distinct shear stress threshold was found, below which little or no particle removal occurred.

4:53PM L25.00007 Large-Scale Simulations of Realistic Fluidized Bed Reactors using Novel Numerical Methods , JESSE CAPECELATRO, OLIVIER DESJARDINS, PERRINE PEPIOT, cornell University, NATIONAL RENEWABLE ENERGY LAB COLLABORATION — Turbulent particle-laden flows in the form of fluidized bed reactors display good mixing properties, low pressure drops, and a fairly uniform temperature distribution. Understanding and predicting the flow dynamics within the reactor is necessary for improving the efficiency, and providing technologies for large-scale industrialization. A numerical strategy based on an Eulerian representation of the gas phase and Lagrangian tracking of the particles is developed in the framework of NGA, a high- order fully conservative parallel code tailored for turbulent flows. The particles are accounted for using a point-particle assumption. Once the gas-phase quantities are mapped to the particle location a conservative, implicit diffusion operation smoothes the field. Normal and tangential collisions are handled via soft-sphere model, modified to allow the bed to reach close packing at rest. The pressure drop across the bed is compared with theory to accurately predict the minimum fluidization velocity. 3D simulations of the National Renewable Energy Lab's 4-inch reactor are then conducted. Tens of millions of particles are tracked. The reactor's geometry is modeled using an immersed boundary scheme. Statistics for volume fraction, velocities, bed expansion, and bubble characteristics are analyzed and compared with experimental data.

5:06PM L25.00008 Large-Scale Eulerian-Lagrangian Simulations of Turbulent Particle-Laden Riser Flows , OLIVIER DESJARDINS, JESSE CAPECELATRO, Cornell University — Turbulent gas-particle flows play fundamental roles in a wide range of technical systems. Understanding and predicting particle-laden turbulent flows is key to ensuring optimal performance and improving the design of devices such as fluidized bed reactors. In this work, a Lagrangian description of the particles is combined with state-of-the-art schemes for high-fidelity turbulence simulations in order to enable predictive numerical modeling of particle cluster formation in turbulent riser flows. The simplified riser configuration of He et al is used to answer several key questions regarding meso-scale structures in risers, in particular regarding (1) the onset of instability, especially in the limit of low volume fractions, (2) the role played by the drag model formulation (in particular the dependence of the drag law on void fraction) and (3) the collision model in the formation and dynamics of particle structures. Simulation results are compared with experimental results in terms of cluster size and shape, as well as gas and particle statistics. Then, a wall-free fully periodic configuration is considered and differences in cluster statistics are discussed.

5:19PM L25.00009 Porosity-Permeability Relations in Granular, Fibrous and Tubular Porous Media[1] , FENG XIAO, XIAOLONG YIN, Petroleum Engineering Department, Colorado School of Mines — A Voronoi diagram-based stochastic geometry generator was developed to generate porous media models of granular, fibrous and tubular types. By adjusting geometry parameters such as number of random seeds and width of channels between grains or radius of fibers/tubes, homogenous and isotropic models of porous media with specified porosity can be accurately generated. The relation of porosity to geometry parameters was proven to be repeatable, and additional manipulations on geometries were built in, including creation of anisotropy and heterogeneity. A parallelized Lattice Boltzmann simulator with nearly ideal speedup was developed and employed to study porosity-permeability relations. Simulation data obtained in the porosity range of 0.01-0.4 revealed that properly normalized permeability in tubular porous media is higher than that in the granular type when porosity becomes greater than 0.1, which can be explained by its more efficient use of the pore space to conduct the flow. Simulation data obtained from fibrous media in solid volume fraction range of 0.01-0.4 agreed with published results, and showed a rapid change with solid volume fraction in the dilute limit.

[1]This research is supported by Research Partnership to Secure Energy for America (RPSEA) 09122-29.

5:32PM L25.00010 Modeling enduring contact in Direct Numerical Simulations of incipient motion of granular beds , JULIAN SIMEONOV, JOSEPH CALANTONI, Naval Research Laboratory — A boundary integral method for fast Direct Numerical Simulations of particle-laden flow on Cartesian grids is used here to model the incipient motion of particle beds forced by a steady current. The particle hydrodynamic force is determined numerically by resolving the flow around individual particles. Analytical lubrication force corrections are added during collisions when the interstitial gap becomes smaller than the grid step. The mechanical force normal and tangential to the contact between particles is modeled by a linear elastic-plastic law and a history dependent friction law, respectively. Resolving fluid-structure interaction effects during collisions is numerically expensive because the collision time scale is two orders of magnitude smaller than the maximum time step for numerical stability of the viscous flow. We improve the numerical efficiency with an acceleration-based control of the time step so that the maximum time step is used for enduring contacts with low particle acceleration. Quantitative comparison for sediment motion initiation is made with laboratory data.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L26 Minisymposium: Cardiac Fluid Dynamics: Translating Fundamental Insights into Clinical Practice 329

3:35PM L26.00001 How the heart works when it fills: what every fluid mechanician needs to know , SANDOR J. KOVACS[1], Cardiovascular Biophysics Laboratory, Washington University School of Medicine, St Louis, MO 63110 USA — The two principles that govern the diastolic (filling) phase of all human hearts are: "constant volume pump" and "suction pump." The $\approx$ 850 ml volume of the pericardial sack decreases by only $\approx$ 40 ml by end systole. This requires that atrial-ventricular volumes simultaneously reciprocate and it underscores the pressure pump (systolic) and volume pump (diastolic) roles of the chambers. Of the 4 heart chambers – ONLY the left ventricle actually serves as a systolic pressure pump. When the normal left ventricle initiates filling after mitral valve opening, it generates only a small (4mmHg) maximum atrioventricular pressure gradient (LVP$<$ LAP) while its pressure continues to decrease for about 100 msec while its volume increases ($dP/dV < 0$). Because the chamber recoils faster than it can fill it is a suction (volume) pump. The purpose of diastole is to fill the chamber (mass transfer) in the fraction of a second available in order to maintain cardiac output. The streamlines entering through the 5cm^2 mitral valve initially have a blunt velocity profile and because mitral valve plane alignment is off-center relative to LV long axis, blood rapidly forms an asymmetric toroidal vortex whose formation time has been shown to depend on LV chamber parameters of stiffness, relaxation and load. Recent Lagrangian coherent structure (LCS) analysis of vortex ring growth in the LV reveals nature's elegant fluid mechanics based solution to the diastolic mass transfer problem. The intraventricular vortex also "rinses" the trabeculated inner surface of the heart thereby preventing formation of blood clots and facilitates mitral leaflet coaptation to minimize mitral valve regurgitation.

[1]Professor of Medicine and Physiology, Adjunct Professor of Physics and Biomedical Engineering

4:01PM L26.00002 Hypertrophic Cardiomyopathy (HCM): How Flow Analysis May Drive Medical Management and Surgical Approach

, THEODORE P. ABRAHAM[1], Translational Cardiovascular Ultrasound Laboratory, Johns Hopkins University School of Medicine, Baltimore, MD USA — Hypertrophic Cardiomyopathy (HCM) is the most common inherited heart disease and occurs in 1 in 500 persons worldwide regardless of race, age and gender. It is the most common cause of sudden death in the young and also causes heart failure and cardiac arrhythmias. The primary anatomic abnormality is thickening of certain walls, or sometimes global thickening of the left or right ventricle. The patterns of thickening along with increased ventricular stiffness lead to suboptimal ventricular filling and inefficient ejection of blood from the ventricle. Treatment for HCM can be medical or surgical. The choice of therapy is driven by the presence and severity of outflow obstruction. Flow analysis could provide sophisticated information about outflow and inflow ventricular dynamics. These flow dynamics features may enable better medical choices and provide information that would allow superior surgical planning.

[1]Associate Professor of Medicine & Director, Hypertrophic Cardiomyopathy Clinic

4:27PM L26.00003 Vortices formed on the mitral valve tips aid normal left ventricular filling

, PAVLOS VLACHOS[1], Department of Mechanical Engineering, Virginia Tech., Blacksburg VA, USA — For the left ventricle to function as an effective pump it must be able to fill from a low left atrial pressure. However, this ability is lost in patients with heart failure. We investigated the fluid dynamics of the left ventricle filling by imaging the blood flow in patients with healthy and impaired diastolic function, using 2D phase contrast magnetic resonance imaging and we quantified the intraventricular pressure gradients and the strength and location of the formed vortices. We found that during early filling in normal subjects, prior to the opening of the mitral valve the flow moves towards the apex and subsequently at the time of the opening of the valve the rapid movement of the mitral annulus away from the left ventricle apex enhances the formation of a vortex ring at the mitral valve tips. Instead of being a passive byproduct of the process as was previously believed, this vortex ring facilitates filling by reducing convective losses and enhancing the function of the left ventricle as a suction pump. Impairment of this mechanism contributes to diastolic dysfunction, with the left ventricle filling becoming dependent on left atrial pressure, and eventually leading to heart failure.

[1]John R. Jones Professor

4:53PM L26.00004 Multiscale modeling and surgical planning for single ventricle heart patients

, ALISON MARSDEN, Mechanical and Aerospace Engineering, University of California San Diego, CA, USA — Single ventricle heart patients are among the most challenging for pediatric cardiologists to treat, and typically undergo a palliative course of three open-heart surgeries starting immediately after birth. We will present recent tools for modeling blood flow in single ventricle heart patients using a multiscale approach that couples a 3D Navier-Stokes domain to a 0D closed loop lumped parameter network comprised of circuit elements. This coupling allows us to capture the effect of changes in local geometry, such as shunt sizes, on global circulatory dynamics, such as cardiac output. A semi-implicit numerical method is formulated to solve the coupled system in which flow and pressure information is passed between the two domains at the inlets and outlets of the model. A finite element method with outflow stabilization is applied in the 3D Navier-Stokes domain, and the LPN system of ordinary differential equations is solved numerically using a Runge-Kutta method. These tools are coupled via automated scripts to a derivative-free optimization method. Optimization is used to systematically explore surgical designs using clinically relevant cost functions for two stages of single ventricle repair. First, we will present results from optimization of the first stage Blalock Taussig Shunt. Second, we will present results from optimization of a new Y-graft design for the third stage of single ventricle repair called the Fontan surgery. The Y-graft is shown, in simulations, to successfully improve hepatic flow distribution, a known clinical problem. Preliminary clinical experience with the Y-graft will be discussed.

5:19PM L26.00005 Cardiac Hemodynamics in the Pathogenesis of Congenital Heart Disease and Aortic Valve Calcification

, VISHAL NIGAM[1], Department of Pediatrics, Section of Cardiology, CA, USA University of California San Diego School of Medicine — An improved understanding of the roles of hemodynamic forces play in cardiac development and the pathogenesis of cardiac disease will have significant scientific and clinical impact. I will focus on the role of fluid dynamics in congenital heart disease and aortic valve calcification. Congenital heart defects are the most common form of birth defect. Aortic valve calcification/stenosis is the third leading cause of adult heart disease and the most common form of acquired valvular disease in developed countries. Given the high incidence of these diseases and their associated morbidity and mortality, the potential translational impact of an improved understanding of cardiac hemodynamic forces is very large.

[1]Division of Pediatric Cardiology, Rady Children's Hospital, San Diego

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L27 Minisymposium: The Fluid Dynamics of Geological CO2 Sequestration Ballroom I-II

3:35PM L27.00001 Fluid dynamics of CO_2 sequestration

, HERBERT E. HUPPERT, Institute of Theoretical Geophysics, University of Cambridge, UK — A means of reducing environmental damage due to anthropogenic emissions of carbon dioxide (CO_2) is through geological storage in porous reservoir rocks until well past the end of the fossil fuel era. Here we discuss the propagation and form of the buoyancy-driven propagation of multiphase CO_2-brine plumes bounded by an impermeable barrier or cap rock. Long-term containment of CO_2 is important, and we will quantify some of the risks due to leakage in this system. Finally, stable sequestration through capillary forces or through dissolution of CO_2 into the brine is greatly enhanced by mixing, which is often dominated by layered stratigraphy. Here we describe injection into a two-layered porous medium, and show the sensitive dependence of propagation and mixing on the input flux, Q. For two-layered systems we find that above a critical flux, Q_C, fluid injected at the base of a relatively low permeability layer preferentially flows in the more permeable upper layer leading to an overriding current, thus enhancing mixing. Finally, we apply these ideas to examine the storage of CO_2 within the Sleipner field, where CO_2 has been injected since 1996. The talk will be illustrated by some desktop experiments.

4:01PM L27.00002 Monitoring pressure evolution during geological CO_2 storage

, MARC HESSE, Geological Sciences and Institute for Computational Science and Engineering, University of Texas at Austin, TX — Pressure build-up near the injection well is a critical factor limiting injection rates during CO_2 storage and leads to measurable deformation at the surface above the injection site. The radial solutions for pressure and saturation in two-phase compressible flow are self-similar and they illustrate that the pressure outside the two- phase region is comparable to single-phase flow. However, pressure dissipation into ambient rocks reduces lateral pressure propagation significantly. Pressure build-up also leads to surface deformation and provides a monitoring tool to invert for reservoir parameters. We formulate an inverse problem to infer the permeability distribution in a quasi-static poroelastic model. Here, we neglect two-phase flow and focus on pressure dissipation into ambient formations. The misfit between model and observations is minimized under the constraint of the poroelastic equations. A numerical study of injection into a deep layer illustrates the possibilities and limitations of retrieving lateral permeability variations from a coupled inversion.

4:27PM L27.00003 CO_2 migration and sequestration by combined capillary and solubility trapping: theory, experiments, and capacity estimates at the basin scale , RUBEN JUANES, Department of Civil and Environmental Engineering, Massachusetts Institute of Technology, Cambridge MA — The large-scale injection and storage of carbon dioxide (CO_2) into deep saline aquifers is a promising tool for reducing atmospheric CO_2 emissions to mitigate climate change. Success of geologic sequestration relies on trapping the buoyant CO_2, to minimize the risk of leakage into shallower formations through pre- existing wells, fractures or faults. However, traditional reservoir-simulation tools are currently unable to resolve the impact of small-scale trapping processes on fluid flow at the scale of a geologic basin. Here, we formulate a sharp-interface mathematical model for the post-injection migration of a CO_2 plume driven by groundwater flow in a sloping aquifer, subject to both capillary trapping and CO_2 dissolution by convective mixing. We develop semi-analytical solutions that elucidate the nontrivial interplay between the two trapping mechanisms, and how their synergetic action controls plume migration. We validate the theory by means of laboratory experiments with analogue fluids to study how convective mixing arrests the buoyant current. We use our findings to estimate the dimensionless rate of solubility trapping for several large saline aquifers in the United States, and assess the importance of solubility trapping in practice.

4:53PM L27.00004 Multiphase, multicomponent simulations and experiments of reactive flow, relevant for combining geologic CO_2 sequestration with geothermal energy capture , MARTIN O. SAAR, Department of Earth Sciences, University of Minnesota - Minneapolis, MN — Understanding the fluid dynamics of supercritical carbon dioxide (CO_2) in brine-filled porous media is important for predictions of CO_2 flow and brine displacement during geologic CO_2 sequestration and during geothermal energy capture using sequestered CO_2 as the subsurface heat extraction fluid. We investigate multiphase fluid flow in porous media employing particle image velocimetry experiments and lattice-Boltzmann fluid flow simulations at the pore scale. In particular, we are interested in the motion of a drop (representing a CO_2 bubble) through an orifice in a plate, representing a simplified porous medium. In addition, we study single-phase/multicomponent reactive transport experimentally by injecting water with dissolved CO_2 into rocks/sediments typically considered for CO_2 sequestration to investigate how resultant fluid-mineral reactions modify permeability fields. Finally, we investigate numerically subsurface CO_2 and heat transport at the geologic formation scale.

5:19PM L27.00005 Practical applications of CO_2 flow modeling in commercial scale sequestration projects , ERNST A. VAN NIEROP, C12 Energy, Berkeley CA, U.S.A. — We review various challenges related to modeling of CO_2 flow through porous media, in the specific context of commercial scale sequestration projects of multiple millions of tons per year. Proper understanding and modeling of the physics of rock- CO_2 and rock-brine interactions have dramatic implications for CO_2 plume spread, and the final "fate" of the injected CO_2. We demonstrate the practical relevance of these concepts on specific geologic sites that are currently being developed for commercial scale sequestration in the United States.

Monday, November 21, 2011 3:35PM - 5:45PM –
Session L29 Low Reynolds Number Swimming in Shear Flow Ballroom III-IV

3:35PM L29.00001 Statics and dynamics of planar shearable filaments in viscous fluid: A Cosserat rod approach , HERMES GADELHA, EAMONN GAFFNEY, ALAIN GORIELY, Mathematical Institute, University of Oxford — Cilia and flagella are ubiquitous in biology as a means of motility and constitute one of the most incredible engineering works of nature. Their inner core, namely the axoneme, consists of a remarkable phylogenetically conserved cytoskeletal structure formed by an assembly of semifexible filaments interconnected by crosslinking proteins. As a result, the flagellum or cilium is not only capable to flexure under the action of an external load, but also to shear. The latter is however a consequence from the intricate elastic crosslinking proteins which causes the elastic bending to couple with shearing deformations, modifying dramatically the effective mechanical response of these bundles of filamentous polymers. We consider deformations of nonlinearly elastic slender rods immersed in a fluid, and analyse the differences between the elastic cross-link shear response and pure material shear resistance under the action of viscous dissipation. We show that pure material shearing effects from Timoshenko's beam theory or, equivalently, Cosserat Rod Theory are fundamentally different from elastic crosslink induced shear found in filament bundles, such as the axoneme.

3:48PM L29.00002 Drag reduction of a hairy disk , JUN NIU, Peking University, DAVID HU, Georgia Institute of Technology — We investigate experimentally the hydrodynamics of a hairy disk immersed in a two-dimensional flowing soap film. Drag force is measured as a function of hair length, density and coating area. An optimum combination of these parameters yields a drag reduction of 17%, which confirms previous numerical predictions of 15% by Favier et al (2009). Flow visualization indicates the primary mechanism for drag reduction is the bending, adhesion and reinforcement of hairs trailing the disk, which reduces wake width and traps "dead water." Thus, the use of hairy coatings can substantially reduce an object's drag while negligibly increasing its weight.

4:01PM L29.00003 Low Reynolds number swimming in a stratified fluid[1] , AMIN DOOSTMOHAMMADI, University of Notre Dame, ROMAN STOCKER, Massachusetts Institute of Technology, AREZOO ARDEKANI, University of Notre Dame — Significant progress has been made in analyzing low-Reynolds number locomotion in homogeneous fluids. Even though many aquatic environments are influenced by vertical variations in density, the effects of stratification on the hydrodynamics of swimming of small organisms are very poorly understood. In this article, by using a squirmer model, we show that motility, energy expenditure, and nutrient uptake of small organisms in a density stratified fluid can be largely influenced due to buoyancy effects. Not only does the stratification suppress the swimming velocity, but it also enhances the nutrient uptake and the energy required for a squirmer to swim across pycnoclines.

[1]This work is supported by NSF grant CBET-1066545.

4:14PM L29.00004 Evaluating Satiated Copepod Behavioral Responses to Thin Layer Flow Structure , AARON C. TRUE, DONALD R. WEBSTER, MARC J. WEISSBURG, JEANNETTE YEN, Georgia Tech — Zooplankton exploit a variety of chemical and fluid mechanical cues in foraging, mate-seeking, and habitat partitioning contexts. To examine the influence of environmental cues on zooplankton aggregations in coastal marine thin layers, a laboratory thin layer mimic was built. The apparatus uses a laminar, planar jet (the Bickley jet) to produce ecologically-relevant layers of chemical (beneficial and harmful phytoplankton) and fluid mechanical (shear strain rate) cues for zooplankton behavioral assays. Particle image velocimetry (PIV) and laser-induced fluorescence (LIF) were employed to fully quantify the spatial structure of the chemical and fluid mechanical cues, ensuring a close match to *in situ* conditions and allowing for investigations into threshold cue levels responsible for inducing behavioral responses. Evaluating the effect of hunger level on behavioral responses is particularly important for producing accurate individual-based simulations of zooplankton population dynamics. Behavioral assays with the calanoid copepod *Temora longicornis* have produced digitized trajectories and, subsequently, path kinematics. Observed behaviors include increased turn frequency and decreased relative swimming speed, which result in increased residence time in the free jet shear layer. Cue-induced individual behaviors have the potential to produce population-scale aggregations.

4:27PM L29.00005 Bacterial Rheotaxis , MARCOS MARCOS, Nanyang Technological University, HENRY FU, University of Nevada, Reno, THOMAS POWERS, Brown University, ROMAN STOCKER, Massachusetts Institute of Technology — Rheotaxis is the directed movement of an organism resulting from fluid velocity gradients, long studied in fish, aquatic invertebrates and spermatozoa. Here we show that rheotaxis also occurs in bacteria. Using controlled microfluidic shear flows, we demonstrate and quantify rheotaxis in *Bacillus subtilis*. A mathematical model of a bacterium swimming in a shear flow is in good agreement with observations and reveals that bacterial rheotaxis results from a subtle interplay between velocity gradients and the helical shape of flagella, which together generate a torque that reorients the cell, altering its swimming direction. The magnitude of the observed rheotactic velocity is comparable to typical chemotactic velocities, suggesting that rheotaxis can interfere with bacterial processes based on directed motility, such as foraging and infection.

4:40PM L29.00006 Bacterial motility and chemotaxis in shear , ROBERTO RUSCONI, JEFFREY S. GUASTO, KWANGMIN SON, ROMAN STOCKER, Massachusetts Institute of Technology — Bacteria often exhibit directed motility ("taxis") in response to gradients of dissolved resources, like nutrients or oxygen. While we have a detailed understanding of chemotaxis in quiescent environments, it has been largely overlooked how this behavior is affected by fluid flow, despite the ubiquity of flow in bacterial habitats. Here we present experiments on aerotaxis (attraction to dissolved oxygen) of *Bacillus subtilis* in controlled shear flows. Using novel microfluidic devices we expose bacterial suspensions to steady oxygen gradients, with independent control over shear rates. From single-cell trajectories and the spatial distribution of bacteria, we show that the cell rotation induced by shear reduces the aerotactic performance, demonstrating that hydrodynamic conditions affect bacterial fitness.

4:53PM L29.00007 Chemotactic Motility of Sperm in Shear , JEFFREY S. GUASTO, MIT, JEFFREY A. RIFFELL, University of Washington, RICHARD K. ZIMMER, UCLA, ROMAN STOCKER, MIT — Chemical gradients are utilized by plants and animals in sexual reproduction to guide swimming sperm cells toward the egg. This process ("chemotaxis"), which can greatly increase the success of fertilization, is subject to interference by fluid flow, both in the bodily conduits of internal fertilizers (e.g. mammals) and in the aquatic environment of external fertilizers (e.g. benthic invertebrates). We studied the biomechanics of chemotaxing sea urchin spermatozoa using microfluidic devices, which allow for the precise and independent control of attractant gradients and fluid shear. We captured swimming trajectories and flagellar beat patterns using high-speed video-microscopy, to detect chemotactic responses and measure the effect of fluid forces on swimming. This work will ultimately help us to understand how swimming sperm cells actively navigate natural chemoattractant gradients for successful fertilization.

5:06PM L29.00008 Enhanced vertical mixing by grass shrimp (Palaemenets) in a linearly stratified fluid , CLAUDIA CENEDESE, Woods Hole Oceanographic Institution, COLM-CILLE P. CAULFIELD, BPI & DAMTP, U. of Cambridge, WILLIAM K. DEWAR, Florida State University, PETER H. WIEBE, NANCY COPLEY, Woods Hole Oceanographic Institution — Laboratory experiments investigated the vertical mixing generated by grass shrimp (Palaemenets) in a linearly stratified tank (50x50x70 cm). The linear stratification ($N = 0.19$ s^{-1}) becomes homogeneous in approximately two days, much faster than if only molecular diffusion was at play. The density evolution in time agrees well with the solution of the diffusion equation for values of enhanced k_s (2.0 10^{-2}cm^2/s when 100 shrimp are present and $k_s = 1.0$ 10^{-2}cm^2/s with 50 shrimp) when compared to the molecular diffusivity of salt $k_s = 1.3$ 10^{-5} cm^2/s. Hence, the laboratory experiments suggest that the enhanced vertical mixing generated by the presence of grass shrimp in a linearly stratified ambient fluid has a diffusive behavior with a diffusivity enhanced by 3 orders of magnitude when compared to the molecular diffusivity of salt.

5:19PM L29.00009 Bacterial locomotion in a wall bounded shear flow , JIAN SHENG, Aerospace Engineering and Mechanics, University of Minnesota, ROMAN STOCKER, Massachusetts Institute of Technology, MEHDI MOALEI, Aerospace Engineering and Mechanics, University of Minnesota — Statistically robust experimental observations on swimming characteristics of bacteria in a wall bounded shear flow are crucial for understanding cell attachment and detachment, interfacial rheology during the initial formation of a biofilm. We combined microfluidics and holography to measure 3-D trajectories of *E. coli* (AW405), subjecting to a carefully controlled shear flow. Experiments are conducted in a straight micro-channel of 40x3x0.2 mm, latter being the depth, with the maximum shear rates up to 200 s^{-1}. Holographic microscopic movies recorded at 40X magnification and 15 fps are streamed in real time to a data acquisition computer for an extended period of time (>20 min) that allows us to examine long term shear responses. Three-dimensional locations and orientations of bacterial are extracted with a resolution of 0.185 μm in lateral directions and 0.5 μm in the wall normal direction. The 3-D trajectories are tracked by an in-house developed particle tracking algorithm. Over thousand 3-D trajectories over a sample volume of 380×380×200 μm have been obtained. On-going analysis focuses on the effects of flow on cell migration and attachment near a sheared surface.

5:32PM L29.00010 Bacteria Aggregation in a Steady Vortical Flow , SHAHRZAD YAZDI, SIXING LI, TONY JUN HUANG, The Pennsylvania State University, AREZOO ARDEKANI, University of Notre Dame — The interaction between microorganisms and flow field is an important, yet complicated topic that affects the design of biological reactors, marine ecological processes, and biofilm formation in porous media. Vortical structures and secondary flows are inherently present in porous media despite small Reynolds numbers. Our experimental results show that bacteria in a steady vortical flow aggregate and subsequently form biofilm streamers in a microfluidic system. The combined effects of shape, motility and the vortical background flow contribute to this fast bacteria aggregation.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M1 Geophysical Flows: General I 301

8:00AM M1.00001 On the dynamics of homogeneous turbulence near a surface , OSCAR FLORES, Universidad Carlos III, JAMES J. RILEY, University of Washington — It is becoming increasing clear that stably-stratified flows can support a stratified turbulence $k^{-5/3}$ inertial range, different from Kolmogorov's. Stratification inhibits vertical motions, but the large-scale quasi-horizontal motions produce strong vertical shearing and small-scale instabilities. The result is a $k^{-5/3}$ horizontal spectrum for the horizontal velocities at scales larger than the Ozmidov scale, the largest scale that can overturn. For smaller scales, the classical Kolmogorov $k^{-5/3}$ applies. Inspired by data taken near the water surface in a tidal river,[1] we here explore to what extent the dynamics of the nonlinear spectral energy transfer of near-surface turbulence with no mean shear (i.e., horizontally isotropic turbulence bounded by free-slip and no-slip surfaces) is analogous to stably stratified turbulence. To that end, we perform DNS of decaying isotropic turbulence with $Re_\lambda \sim 100$, but bounded by a non-slip surface and a free slip surface. The behavior of the flow near the free-slip surface is similar to stratified turbulence, with a tentative $k^{-5/3}$ range, but the same is not true for the no-slip surface at the present Reynolds numbers. This research was supported by ARO and NSF.

[1]Chickadel et al. (2011) to appear in *IEEE Geosci. Remote Sens. Lett.*

8:13AM M1.00002 The energetics of stably stratified turbulence in the Boussinesq approximation

, SEUNGBUM JO, KEIKO NOMURA, JAMES ROTTMAN, University of California, San Diego — There has been a recent resurgence of interest in determining the consistency of the Boussinesq approximation to describe the coupling of the dynamics and thermodynamics of turbulent stratified flows. In particular, there is some debate over how energy is converted from internal to mechanical energy in this approximation. To gain some insight into these issues, we derive the evolution equations of the different forms of energy for Boussinesq stratified flows from the Lagrangian point of view. This analysis allows us better physical insight into these issues and allows us to show explicitly how energy is converted between internal and mechanical energy. The physical significance of these results will be discussed.

8:26AM M1.00003 Dynamics and structure of stably stratified turbulence

, SIMON SCHAAD, SUBHAS VENAYAGAMOORTHY, Colorado State University, DEREK STRETCH, University of KwaZulu-Natal, South Africa — The structural features of stratified turbulence and its relationship to the flow dynamics has been the subject of many recent investigations. In strongly stratified turbulent flows, the formation of large-scale quasi-horizontal vortices in layers with strong vertical variability has been observed in laboratory experiments. In this study, direct numerical simulations (DNS) of stably stratified turbulence are used to investigate the evolution of flows in terms of overturns and their relationship to mixing. Isosurfaces of enstrophy of strongly stable flows indicate the emergence of randomly distributed "pancake-like" structures with near horizontal orientation at later times. The vertical dynamics of such strongly stratified flows are dominated by linear internal waves and can be described using rapid-distortion theory (RDT) while their horizontal dynamics are dominated by nonlinear effects that cannot be described by RDT. This suggests a decoupling of the vertical and horizontal dynamics of the flow. The "pancake" enstrophy structures appear to be associated with shear layers between layered vortex modes. It has been suggested by others that these shear layers could become locally unstable leading to additional source of turbulence in these strongly stable flows.

8:39AM M1.00004 Energy transfer for stably stratified turbulence

, YOSHIFUMI KIMURA, Nagoya University, JACKSON HERRING, NCAR — To understand the mechanism of producing the power-law transition in the energy spectra for stably stratified turbulence, we investigate the energy transfer using direct numerical simulations (DNS) at a resolution of 1024^3. The calculation is done by solving the 3D Navier-Stokes equations with horizontal forcing under the Boussinesq approximation pseudo-spectrally. Using toroidal-poloidal decomposition (Craya-Herring decomposition), the velocity field is decomposed into vortex and wave modes. We have observed that the vortex and wave spectra are consistent with a Kolgomorov–like $k^{-5/3}$ range at sufficiently large k, and that at large scales, the wave spectrum is a steeper $k_\perp^{-2}$, while that for the vortex component is consistent with $k_\perp^{-3}$ for sufficiently strong stratification. Here $k_\perp$ is the horizontally gathered wave numbers. Therefore there is a power-law transition in the spectra of the both modes. By looking at the energy budget of the wave and vortex modes, we study the energy transfer functions. We will demonstrate that the energy transfer shows a difference from the conventional Kolmogorov picture for isotropic turbulence, in particular at the large scale behavior in which the energy transfer is balanced with enhanced dissipation presumably because of the rough layer edges suggested by Herring & Métais (1989).

Herring, J.R. & Métais, O. : Numerical experiments in forced stably stratified turbulence *J. Fluid Mech.* **202** 97–115 (1989).

8:52AM M1.00005 Kinetic Energy Dynamics in Forced, Horizontally Homogeneous, Stably Stratified Turbulence

, STEVE DE BRUYN KOPS, KAUSTUBH RAO, SABA ALMALKIE, Univ. of Massachusetts Amherst — Recent numerical simulations and scaling arguments have established that in stratified turbulence there is net downscale transfer of kinetic energy. The nature of the transfer is less clear, particularly in how it compares to energy transfer in the inertial range of isotropic turbulence, and the literature suggests that the dynamics depend on the buoyancy Reynolds number, Re_b. To gain further insight, three-dimensional direct numerical simulations with 170 billion grid points are considered. The flows are horizontally homogeneous and vertically stratified with Froude number between 0.125 and 1 and Re_b between 9 and 219. The complete balances of the horizontal and vertical contributions to kinetic energy are presented in terms of two-dimensional spectra. In this format, the extent in wave number space of the inertial range where the dissipation rate is small compared with other terms in the balance is clear. The spectra also show regions of wave space in which energy transfers downscale in the vertical and then upscale in the horizontal, particularly at low Re_b. They also explain why some published results reporting one-dimensional spectra appear to show high dissipation rate at large length scales.

9:05AM M1.00006 Transition to turbulence in homogeneous and stratified Ekman boundary layers - an experimental study[1]

, MANIKANDAN MATHUR, THOMAS DUBOS, Laboratoire de Meteorologie Dynamique, Ecole Polytechnique, 91128 Palaiseau, France, SAMUEL VIBOUD, HENRI DIDELLE, JOEL SOMMERIA, Laboratoire des Ecoulements Geophysiques et Industriels, UMR 5519 CNRS-UJF-INPG, 21 rue des Martyrs, 38000 Grenoble, France — A dynamical interplay between the pressure gradient, the Coriolis force and the viscous forces describes the laminar regime of the (Ekman) boundary layers in the ocean and the atmosphere. Early experiments on the homogeneous Ekman layer identified two branches of instabilities that result in roll patterns with distinct wavelengths and orientation. In this talk, we present results based on quantitative velocity field measurements in experiments performed on the 13m diameter Coriolis rotating platform at Grenoble, France. Geostrophic flow outside the boundary layer is set up in a well-controlled manner by changing the tank rotation appropriately. In the homogeneous case, each of the two branches of instabilities is captured in isolation by abruptly increasing the Reynolds number from 0 to a finite value ranging from 50 to 360. The corresponding wavelengths and growth rates as a function of the Reynolds number are deduced from our measurements. Emission of inertial waves is detected for certain parameters. Finally, the influence of a background stratification on the roll patterns of instability is investigated in some exploratory experiments.

[1]Funding from the research project ANR TRANSTEK is gratefully acknowledged.

9:18AM M1.00007 Analysis of cascades in space and in scale for rotating and stratified Boussinesq flows

, SUSAN KURIEN, HUSSEIN ALUIE, Los Alamos National Laboratory — We use high-resolution simulations of Boussinesq flows, forced in the large-scales, with fixed rotation and stable stratification along the vertical axis, to study the downscale cascades of energy and potential enstrophy in three different regimes of stratification and rotation. (1) For strongly stratified flows with moderate rotation, we observe anisotropic fluxes of energy and potential enstrophy into Fourier modes with large vertical component k_z, predominantly due to a highly non-local transfer from the large-scales directly to the smallest scales. The energy cascade is predominantly due to three vortical-mode interactions. (2) For strongly rotating flow with moderate stratification, there are anisotropic fluxes to modes with large k_h, due to a "diffusely" local transfer much like in isotropic Navier-Stokes turbulence. The energy cascade is primarily due to three vortical-mode interactions, as in the strongly stratified case, although wave-vortical-wave and vortical-wave-vortical interactions also make a noticeable contribution. (3) In the third case of equally strong rotation and stratification, there are only slightly anisotropic fluxes, mostly to modes with large k_h, due to an ultra-local transfer in which the energy gained by an inertial scale comes almost exclusively from the adjacent larger scales. We confirm that the cascades in the third regime are primarily due to wave-vortical-wave interactions, in agreement with previous work.

9:31AM M1.00008 3D Structure and Internal Circulation of Pancake Vortices in Rotating Stratified Flows , PEDRAM HASSANZADEH, PHILIP MARCUS, UC Berkeley, ORIANE AUBERT, MICHAEL LE BARS, PATRICE LE GAL, IRPHE, France — Jovian vortices, Atlantic meddies, and vortices of the protoplanetary disks are examples of weakly-forced or unforced long-lived vortices in rotating stratified flows. Knowing the 3D structure and internal circulation of these vortices is essential in understanding their physics, which is not well-understood. For example, the aspect ratio of these vortices has been long thought to be f/N where f is the Coriolis parameter and N is the Brunt-Vaisala frequency. However, our recent theoretical and experimental study has shown that the aspect ratio in fact depends not only on f and N but also on the Rossby number and density mixing inside the vortex. The new scaling law also agrees with the available measurements of the meddies and Jupiter's Great Red Spot. High resolution 3D numerical simulations of the Navier-Stokes equation are carried out to confirm this new scaling law for a slowly (viscously) decaying anticyclonic vortex in which the Rossby number and stratification inside the vortex evolve in time. For a wide range of parameters and different distributions of density anomaly, the secondary circulations within the vortices are studied. The effect of a non-uniform background stratification is investigated, and the small cyclonic vortices that form above and below the anticyclone are studied.

9:44AM M1.00009 New equations for the slow dynamics in the presence of strong rotation and weak stratification[1] , BETH WINGATE, Los Alamos National Laboratory, PEDRO EMBID, University of New Mexico — We explore the strong rotation limit of the rotating and stratified Boussinesq equations with periodic boundary conditions when the stratification is weak. This regime corresponds to the limit where (Rossby number) $Ro = \epsilon$, (Froude number) $Fr = O(1)$, as $\epsilon \to 0$. We show that the slow dynamics decouples from the fast and derive new equations for the slow dynamics. The new equations for the slow dynamics describe the dynamics of Taylor-Proudman flows and their conservation laws including a new conserved quantity for vertical kinetic energy and potential energy. The leading order potential enstrophy is slow while the leading order total energy retains both fast and slow dynamics. We also perform forced numerical simulations of the rotating Boussinesq equations to demonstrate support for the theory.

[1]Thank you for support from the DOE office of Advanced Scientific Research and the DOE office of biological and environmental resesarch.

9:57AM M1.00010 ABSTRACT WITHDRAWN —

Tuesday, November 22, 2011 8:00AM - 10:10AM —
Session M2 Granular Flows III 302

8:00AM M2.00001 Granular convection in a spherical tumbler[1] , ZAFIR ZAMAN, Northwestern University, UMBERTO D'ORTONA, Université Aix-Marseille, CNRS, JULIO M. OTTINO, RICHARD M. LUEPTOW, Northwestern University — We have performed DEM simulations and experiments for monodisperse particles in partially-filled spherical tumblers to better understand the flow of bidisperse particles in spherical tumblers, which display segregation patterns that result from differential axial migration of the two species. Particle tracking in DEM simulations for a 30% full tumbler suggests that there is a convection pattern in the monodisperse system that could play a role in the bidisperse system. Particles near the surface of the flowing layer move slowly toward the poles while particles lower in the flowing layer move toward the equator. The convection is quite slow, taking O(100) passages through the flowing layer to complete one convection orbit. Experiments in an 8 cm diameter spherical tumbler with 1 mm monodisperse particles having a band of particles of a different color confirm the convection pattern.

[1]Partially supported by NSF grant number 1000469.

8:13AM M2.00002 Instabilities in a Freely Cooling Granular Gas: A Quantitative Comparison of DEM simulations and Kinetic-Theory-based models , PETER MITRANO, Chemical and Biological Engineering, University of Colorado at Boulder, Boulder, CO, ANDREW HILGER, CHRISTINE HRENYA — Experiments, discrete element method (DEM) simulations, and kinetic-theory-based predictions have demonstrated the existence of clustering instabilities in flows of solid, inelastic grains. Such instabilities have also been studied via stability analyses of the continuum balances for rapid granular flows. Spurred by discrepancies between DSMC-based and kinetic-theory-based transport coefficients in extremely dissipative systems, previous work has shown that a modified Sonine approximation reduces this disagreement. However, the quantitative accuracy of this modified kinetic theory with respect to predicting instabilities has not been addressed. In this work, hard-sphere, event-driven DEM simulations of the homogenous cooling system are used to study instabilities in granular systems. Detection of instabilities is determined via Fourier analysis. The aim is to determine the critical system size at which clustering appears over a wide range of dissipation. By comparing the critical size from DEM with the predictions based on standard (Garzó 2005) and modified (Garzó 2007) Sonine approximations, this work aims to assess the quantitative ability of each model to predict instabilities for monodisperse granular flows.

8:26AM M2.00003 Sudden Chain Energy Transfer Events in Vibrated Granular Media , RODRIGO SOTO, Universidad de Chile, NICOLAS RIVAS, SUOMI PONCE, BASILE GALLET, DINO RISSO, PATRICIO CORDERO, NICOLAS MUJICA — In a mixture of two species of grains of equal size but different mass, placed in a vertically vibrated shallow box, there is spontaneous segregation. Once the system is at least partly segregated and clusters of the heavy particles have formed, there are sudden peaks of the horizontal kinetic energy of the heavy particles, that is otherwise small. Together with the energy peaks the clusters rapidly expand and the segregation is partially lost. The process repeats once segregation has taken place again, either randomly or with some regularity in time depending on the experimental or numerical parameters. An explanation for these events is provided based on the existence of a fixed point for an isolated particle bouncing with only vertical motion. The horizontal energy peaks occur when the energy stored in the vertical motion is partly transferred into horizontal energy through a chain reaction of collisions between heavy particles.

8:39AM M2.00004 Packing and stability of geometrically cohesive granular media , NICK GRAVISH, Georgia Tech School of Physics, SCOTT V. FRANKLIN, Rochester Institute of Technology Department of Physics, DAVID L. HU, Georgia Tech School of Mechanical Engineering, DANIEL I. GOLDMAN, Georgia Tech School of Physics — Granular particles with concave shapes may entangle with neighboring particles creating an effective cohesion controlled by particle geometry. We study the packing and stability of vertical columns formed from geometrically cohesive u-shaped particles (staples) of varying barb length, l. We prepare cohesive columns by packing particles in a confining cylindrical tube under vertical vibration at fixed frequency of $f = 30$ Hz and peak acceleration (in units of g) of $\Gamma = 2$. The initial and final volume fraction vary with l and volume fraction increases for decreasing l. Once packed, the tube is removed and columns are subjected to vertical vibration at fixed f and variable Γ. We monitor column height, $h(t)$, during collapse and find that $h(t)$ is described by a stretched exponential $h(t)/h_0 = \exp[-(\frac{t}{\tau})^{\beta}]$. The characteristic collapse time, τ, is governed by an Arrhenius law with $\tau = \tau_0 \exp(\Gamma_0/\Gamma)$ where Γ_0 is a measure of the column's resistance to collapse. We find that Γ_0 is a non-monotonic function of l and exhibits a maximum at intermediate l. We explain this effect through a model considering packing and entanglement.

8:52AM M2.00005 Numerical model for the motion of large object in fluidized bed , TAKUYA TSUJI, KYOHEI HIGASHIDA, TOSHITSUGU TANAKA, Osaka University, OSAKA UNIVERSITY TEAM — The motion of large object in fluidized beds is quite complex because it is influenced by the fluid force, buoyant force in addition to the contact force from emulsion particles, other immersed objects and walls. Its behavior is expected to change depending on its shape, density and size along with the condition of fluidized bed. It is a physically interesting problem and it also has a practical importance in several engineering applications such as gasification, mixing, separation and granulation. In the present study, a numerical model which predicts the motion of large object in fluidized bed is developed. We concentrate on objects larger than surrounding emulsion particles by one order of magnitude at least. The basic concept of a proposed numerical model is presented and its performance is tested for fundamental problems such as sedimentation of single spherical object in a bubbling fluidized bed.

9:05AM M2.00006 Segregation of Binary Mixtures: Competing Effects of Gravity and Shear Rate Gradients , KIMBERLY HILL, University of Minnesota, YI FAN, Northwestern University — Mixtures of particles tend to segregate when they flow. For example, when a dense binary mixture of different-sized particles flows down an inclined plane, the larger particles tend to go up (toward the free surface), and the smaller particles, down. However, this trend is not as simple as it might first seem. It has recently been demonstrated that under otherwise identical conditions the relative segregation fluxes in a binary mixture of particles are not monotonically dependent on particle size ratio. Further, for a single mixture, larger particles may rise or fall relative to the small particles, depending on subtle differences in local flow conditions. We have recently shown that a shear rate gradient drives segregation in dense flows, and that the direction depends on the partitioning of stress between species. The complicated segregation dynamics in gravity-driven flow may be due to competing effects of gravity and shear rate gradients in segregating particles. We combine a theory for gravity-driven segregation of granular materials (Gray and Thornton, Proc. R. Soc. A, 2005) with our theory for shear-driven segregation (New J Phys, in press) to study how these competing segregation effects can give rise to the variety of trends that have been observed.

9:18AM M2.00007 Kinetic theory based computational approaches for granular flows , PRAKASH VEDULA, University of Oklahoma, Norman — Challenges for efficient computational prediction of granular flows arise particularly due to their nonequilibrium behavior, as a result of which the continuum field assumptions become invalid (especially when the effective Knudsen number is not a small parameter). To address some of these challenges, we present computational methods for treatment of granular flows including particle rotation effects, based on solution of the corresponding Boltzmann equation with full collision operator using fixed-lattice and adaptive quadrature based approaches. Nonequilibrium behavior in dilute granular flows consisting of smooth and rough spheres will be investigated. The effects of inelasticity and mass distribution of the constituent spheres will also be discussed. Detailed treatment of the collision operators in our approaches ensures that the collision invariants are preserved and the functional dependence of evolution of generalized moments (involving linear and angular velocity components) is correctly represented. Semi-analytic representations of the moment contributions due to collision operator, which involve fourteen dimensional integrals, will also be developed. Microscopic dynamics of binary collisions is considered to relate pre-collisional and post-collisional linear and angular velocities of particles.

9:31AM M2.00008 Numerical study of unsteady shear, and gravity-driven granular flows , CHRISTOS VARSAKELIS, MILTIADIS V. PAPALEXANDRIS, Universite catholique de Louvain — In this talk, we report on the results of a numerical study for unsteady, shear-driven granular flows and granular flows over inclined planes. The numerical simulations have been performed with the application of a new algorithm for two-phase continua that has been recently developed by our team. After a short description of the algorithm, we present representative results of our numerical simulations. Further, we discuss the characteristics of the emerging flow structures and their dependence on certain physical parameters of the granular phase, such as, initial concentration, viscosity coefficient, etc. The talk concludes with a comparison of our numerical predictions at large times with those of previous studies on steady granular flows.

9:44AM M2.00009 Layer Depth Dependence of Shocks and Patterns in Shaken Granular Systems[1] , MICHAEL HOLLOWED, JON BOUGIE, Loyola University Chicago — We simulate shaken granular layers using numerical solutions of continuum equations to Navier-Stokes order and use these simulations to study shocks and patterns in vertically oscillated layers of grains. When the accelerational amplitude of the forced oscillations exceeds that of gravity, the layer leaves the plate at some time during the cycle. When the grains collide with the plate later in the cycle, shocks are formed within the layer. If the accelerational amplitude exceeds a critical value, standing wave patterns also form. We investigate the interactions between shocks and patterns in these systems, as well as their dependence on layer depth. We demonstrate relationships between properties associated with shocks (such as Mach number and pressure gradients across the shock) and properties associated with standing wave patterns (such as pattern wavelength and horizontal flow speed between peaks). Re-scaling these quantities by the depth of the layer yields relationships between dimensionless quantities that are valid across a range of layer depths.

[1]This research was supported by Research Corporation for Science Advancement.

9:57AM M2.00010 The stress profile in a sheared granular column[1] , PRABHU NOTT, Indian Institute of Science, VISHWAJEET MEHANDIA, Universite de Strasbourg, KAMALA JYOTSNA GUTAM, Dr. Reddy's Laboratories — It has been known for several centuries that the normal stress at the base of a column of granular material deviates from the value dictated by the hydrostatic balance. This was explained by Janssen (1895) as being due to the shear stress imposed by the confining walls on the granular column, as a result of grain-wall friction. The question we address in this presentation is, what is the stress field when the column is sheared? Depending on the assumptions on the kinematics, plasticity theories predict that the stress profile is similar either to that in a static column, or to that in a sheared fluid column. Here, we report the results of our experimental study of slow shear of a granular material in a cylindrical Couette cell, in which all components of the stress were measured at the stationary outer cylinder. The stress was measured as a function of distance from the free surface. The results of our experiments are intriguing: the radial normal stress deviates strongly from the predictions of all available theories and previous experimental measurements. The axial shear stress changes sign when a static column is sheared. We describe these results, and speculate as to which type of theory might explain the observations.

[1]Support from the DST, India is gratefully acknowledged.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M3 Bubbles III 303

8:00AM M3.00001 Optical Tomography of Polydisperse Dry Foam[1] , ANTHONY CHIECO, KLEBERT FEITOSA, Dept. of Physics and Astronomy, James Madison University, P.T. KORDA, A.E. ROTH, D.J. DURIAN, Dept. of Physics and Astronomy, University of Pennsylvania — Dry foam is a disordered packing of bubbles that distort into familiar polyhedral shapes. We have implemented a method that uses optical axial tomography to reconstruct the internal structure of a dry foam in three dimensions. The technique consists of taking a series of photographs of the dry foam against a uniformly illuminated background at successive angles. By summing the projections we create images of the cross section of the foam and analyze them to locate the Plateau borders and vertices. The vertices are then connected according to Plateau's rules to reconstruct the internal structure of the foam. Using this technique we are able to visualize a large number of bubbles of real 3D foams and obtain statistics of faces and edges.

[1]We gratefully acknowledge support from DOD-ASSURE/NSF-REU grant # DMR-0851367.

8:13AM M3.00002 Wall effects in Stokes experiment with a liquid foam , HAIJING GAO, Purdue University, HARIPRASAD SUBRAMANI, Chevron Corporation, MICHAEL HARRIS, OSMAN BASARAN, Purdue University — Liquid foams are widely used in numerous applications ranging from the oil and gas industry to beauty, healthcare, and household products industries. A fundamental understanding of the relationships between the properties of liquid foams and their flow responses is, however, still in its infancy compared to that involving the fluid dynamics of simple fluids. In this talk, the flow of a dry liquid foam around a spherical bead, i.e. the Stokes problem for liquid foams, is studied experimentally. In contrast to previous work (cf. Cantat 2006), the focus of the present research is to probe the effect of a solid wall that is located a few bubble radii from the bead. The new experimental results show that the elastic modulus of dry liquid foams is directly proportional to the surface tension of the foaming agents and inversely proportional to the average bubble size in the foams, in agreement with previous theoretical and experimental studies. The experiments further show that the close proximity of the solid wall causes profound structural changes to the gas bubbles as the foam flows past the bead. A good understanding of these structural changes and how they can affect the elastic modulus of foams can be indispensable in formulating improved models for accurately describing the dynamical response of foams within the realm of continuum mechanics.

8:26AM M3.00003 Velocimetry of aqueous foam drainage , MATTHEW J. KENNEDY[1], JAMES W. FLEMING, Naval Research Laboratory — We study the flow behavior of a freely draining aqueous foam using microparticle image velocimetry. Video shows liquid traveling uphill along the bubble surfaces, counter-directional to the primary flow. Most theoretical models of foam drainage avoid direct calculation of the flow at the gas-liquid interface. Rather, they treat the effect of surface flow empirically due to lack of experimental and theoretical understanding of the forces which cause flow at the surface. This uphill flow represents a significant component of the overall flow. Understanding the surface-driven flow can improve predictions of water-retention time. In the present work, we use a proprietary fire-suppression foam solution comprising hydrocarbon-based non-fluorinated surfactants in water. At the start of drainage, the mean upward flow speed is 0.2 $\pm$ 0.1 mm/s while the mean downward flow speed is 0.8 $\pm$ 0.1 mm/s. The local bubble size at the location-under-imaging is 0.6 mm with a coefficient of variation of 50%. Ongoing efforts concern the effect of the liquid solution constituents on the uphill flow and the effect of bubble size on drainage time.

[1]NRC/NRL Postdoctoral Fellow

8:39AM M3.00004 Bubble rearrangements dynamics in foams , MARIE LE MERRER, INSP, Univ. Paris 6, SEVERINE COSTA, SYLVIE COHEN-ADDAD, REINHARD HOEHLER, INSP, Univ. Paris 6 / LPMDI, Univ. Paris Est — Liquid foams are jammed dispersions of gas bubbles in a surfactant solution. Their structure evolves with time because surface tension drives a diffusive gas exchange between neighboring bubbles. This coarsening leads to a build-up of stresses which are relaxed upon local intermittent bubble rearrangements. These events govern the slow viscoelastic foam response, and similar bubble rearrangements are the elementary processes of plastic flow. Thus, the rearrangement duration is a key parameter describing how the microstructure dynamics control the macroscopic rheological response. We probe the duration of coarsening-induced rearrangements in 3D foams using a multiple light scattering technique (time resolved Diffusing-Wave Spectroscopy) as a function of the surfactant chemistry and the liquid fraction. As the foam becomes wetter, the confinement pressure of the packing goes to zero and the contacts between bubbles vanish. For mobile interfaces, we find that the rearrangements slow down as the jamming point is approached. These findings are compared to scaling laws which reveal an analogy between rearrangements dynamics in foams and granular suspensions.

8:52AM M3.00005 Bubbling regime in planar co-flowing air-water sheets[1] , C. GUTIÉRREZ-MONTES, R. BOLAÑOS-JIMÉNEZ, University of Jaen, Spain, A. SEVILLA, University Carlos III of Madrid, Spain, E. SANMIGUEL-ROJAS, C. MARTÍNEZ-BAZÁN, University of Jaen, Spain — We study by means of experiments and numerical simulations the periodic breakup of an air sheet surrounded by a co-flowing water sheet, for a fixed liquid- to-gas thickness ratio, $h = h^*_{w,0}/h^*_{a,0} \simeq 5.27$, as a function of their control parameters, namely the Weber number, $We = \rho_w\, u^{*2}_{w,0}\, h^*_{a,0}/\sigma$, and the water-to-air velocity ratio, $\Lambda = u^*_{w,0}/\bar{u}^*_{a,0}$, where $h^*_{a,0}$ and $h^*_{w,0}$ are the half-thicknesses of the air and water sheets at the exit. The bubble formation process is divided into two stages governed by different physical phenomena: the formation of an incipient neck, which moves downstream at the water velocity, and the collapse of the neck. Both stages are characterized and a simple theoretical model, which depends on the control parameters, is proposed for each one in order to provide the time scale for the bubbling process, finding a good agreement with the experiments and simulations. Finally, the experimental measurements of the bubbling frequency and bubble size are compared with those obtained numerically within the We- Λ parameter space studied.

[1]Supported by the Spanish MICINN, Junta de Andalucía and EU Funds under projects DPI2008-06624-C03-02 and P07-TEP02693.

9:05AM M3.00006 Universality of Bubble-Jets in Gravitational Fields[1] , DANAIL OBRESCHKOW, MARC TINGUELY, NICOLAS DORSAZ, PHILIPPE KOBEL, AURELE DE BOSSET, MOHAMED FARHAT, Ecole Polytechnique Federale de Lausanne — Gravity matters: for us, as well as for small vapor bubbles in liquids. When cavitation bubbles collapse, they feel the presence of a faint hydrostatic pressure gradient caused by gravity – an effect, which is widely neglected for the experimental difficulty of uncovering the weak action of gravity. We faced this challenge by designing an experiment able to generate uniquely spherical cavitation bubbles. Captivating high-speed movies showing the collapse of those bubbles manifest beautiful jets caused solely by gravity. These jets were studied systematically by running the experiment aboard an acrobatic aircraft (52nd ESA parabolic flight campaign), able to simulate the gravitational fields of the smallest moons and the largest planets in the solar system. The data reveals a clear connection between the size of the jets and the level of gravity. Further reduction and theoretical developments led to a universal scaling law between the size of jets emitted by cavitation bubbles and a single parameter, which only depends on the pressure field and the bubble volume.

[1]SNSF Grant 200020-116641

9:18AM M3.00007 Using a polydisperse bubble cloud as a non-invasive pressure probe , ANA MEDINA-PALOMO, JAVIER RODRIGUEZ-RODRIGUEZ, Carlos III Univ. of Madrid, Spain — Along the last 30 years, several techniques have been developed to measure non-invasively the pressure inside a liquid exploiting different features of the acoustic spectrum scattered by a cloud of bubbles. However, no technique is commercially avalilable nowadays, as all of them encounter technical difficulties that preclude their implementation. One of the common problems is that, in real (mostly medical) applications, bubble populations exhibit a wide range of sizes. We have studied numerically the acoustic behavior of polydisperse bubble clouds with the aim at evaluating the accuracy with which the pressure can be measured using one of these techniques. Namely, the determination of the pressure dependent component of the resonance frequency (Minnaert's frequency). A parametric study has been performed varying the properties of individual bubbles as well as those of the bubble distribution. The accuracy of the pressure measured using different conditions is reported, with suggestions for engineers and scientist working on the design of novel microbubbles (such as Ultrasound Contrast Agents). Finally, we have also developed a simplified expression to calculate the acoustic spectrum of a bubble population. This expression allows us to estimate, without the need of full simulations, the effect of the different parameters explored. Supported by the Spanish Ministry of Science through grant DPI2008-06369.

9:31AM M3.00008 Infrared Measurements of Multiphase Flow Phenomena[1] , JUNGHO KIM, TAE HOON KIM, ERIC KOMMER, SERGUEI DESSIATOUN — Understanding of phase change heat transfer mechanisms remains elusive due its sensitivity to many variables, but also due to the lack of reliable local information that can enable models to be tested. Although point measurements of variables such as local film thickness and heat transfer have been made, techniques whereby these quantities can be measured over the large areas have been lacking. IR thermometry is an established technology that can be used where optical access to the surface is available in the wavelength of interest. The use of IR measurements is demonstrated in this work to measure the inner and outer wall temperatures of an electrically heated silicon tube during flow boiling of FC-72. The electrical conductivity of silicon can be varied over a broad range by controlling the dopant concentration. Since silicon is largely transparent to IR radiation, the temperature of the inner and outer walls can be measured by coating selected areas with IR opaque thin films. FC-72 is also partially transparent to IR radiation over a broad range of wavelengths, allowing the flow to be visualized. Details of the proposed technique, test apparatus, data reduction, and model development are presented.

[1]This work was sponsored by NASA HQ Grant NNX09AK39A.

9:44AM M3.00009 Flow of Air Bubbles in a Packed Bed , SOTO ENRIQUE, ZENIT ROBERTO, Universidad Nacional Autonoma de Mexico, AGUILAR-CORONA ALICIA, Universidad Michoacana de San Nicolas de Hidalgo — The flow of bubbles through a granular media is presented. An index matching technique was used to visualize the permeation of bubble inside the packed bed formed by spheres with diameters from 0.5 mm to 6 mm and specific densities of 1.05 and 2.45. The volume of the bubble is determined by the flow rate and four distinctive modes were found. The first is for small bubbles that percolate without any deformation. The second is for larger bubbles that are deformed but still passed through the bed without break up. For the particles with higher density the third mode is observed and consists in the formation of air paths inside the bed. In the last mode, the air is cumulated till the bubble reaches a critical volume. Above the critical volume the granular media is fluidized and the bubble flows freely inside it. During this process some particles are dragged to the liquid bulk. The pore size and the capillary length were measured. Furthermore, the comparison of the bubbles velocity and shape inside the bed and in the liquid bulk are presented. These results agree with others authors who studied the emergence of bubbles from an immersed granular bed.

9:57AM M3.00010 Anomalous coalescence in sheared two-dimensional foam , HADI MOHAMMADIGOUSHKI, GIOVANNI GHIGLIOTTI, Department of Chemical and Biological Engineering, University of British Columbia, Vancouver, BC V6T 1Z3, Canada, G.M. HOMSY, Department of Mathematics, University of British Columbia,Vancouver, BC V6T 1Z2, Canada, JAMES FENG[1], Department of Chemical and Biological Engineering, University of British Columbia, Vancouver, BC V6T 1Z3, Canada — We report an experimental study on shearing a monolayer of monodisperse bubbles floating on liquid in a narrow-gap Couette device. The bubbles in such a "bubble raft" coalesce only if the shear rate exceeds a threshold value. This is in contrast to the conventional wisdom that bubbles and drops coalesce for gentler collisions, at shear rates below a critical value. Furthermore, the threshold shear rate increases with the bubble size and the viscosity of the suspending liquid, contravening reasoning based on capillary number. Through visualization and scaling arguments, we have advanced an explanation of the anomalous behavior in terms of inertial forces on the bubbles, which compress the bubbles radially inward and accelerate film drainage. The scaling relationship correlates well with experimental data.

[1]Department of Mathematics, University of British Columbia,Vancouver, BC V6T 1Z2, Canada

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M4 Drops IX: Spreading on Surfaces 307

8:00AM M4.00001 Spreading and wetting of impacting drops: Three-dimensional simulations using Conformal Decomposition Finite Elements[1] , SCOTT ROBERTS, JEREMY LECHMAN, Sandia National Laboratories — Understanding the spreading and wetting of liquid drops impacting a solid substrate is of interest to many industrial processes, including coating, printing, and thermal spray processes. Depending on the contact angle and wetting behavior, these drops can exhibit many shapes, including disks, crowns, and rebounding drops. Despite its importance, accurately capturing these dynamics using numerical simulations remains a daunting task. In this talk, we use a new Conformal Decomposition Finite Element Method (CDFEM), which seeks to capture the benefits of both moving-mesh and level-set methods, to study drop impact and spreading. Three-dimensional simulations are performed and interface profiles and drop spreading ratios are compared to previous experimental and theoretical studies. The strength of this method is also demonstrated in more complicated geometries, where three-dimensional simulations are necessary.

[1]Sandia is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

8:13AM M4.00002 Slip in viscous contact-line movement , HENRIK VAN LENGERICH[1], Brown University, PAUL STEEN, Cornell University, KENNETH BREUER, Brown University — The typical continuum fluid dynamics formulation cannot be used to model the spreading of a liquid on a solid because a stress singularity prevents contact-line motion. It is well known that this situation can be remedied by introducing a slip. We perform Stokes-flow simulations with slip and compare these with experiments. In the experiment, liquid (squalane) is forced through two parallel sapphire plates (roughness 0.6nm), and the meniscus shape and its speed are measured. The slip-length for this liquid/solid pair has been measured previously in an independent experiment absent of contact lines (T. Schmatko et. al. PRL 94, 244501). The same geometry is used in a boundary integral method simulation, accurate to within a few molecular diameters in the vicinity of the contact-line. The slip-length in the simulations can be varied such that the meniscus shape matches the experiment. Preliminary results suggest this slip-length is an order of magnitude lower than that reported by Schmatko.

[1]Now at the University of Minnesota TC

8:26AM M4.00003 ABSTRACT WITHDRAWN –

8:39AM M4.00004 Spreading and Arrest of Molten Liquid on Solid Substrates , FARYAR TAVAKOLI, PIROUZ KAVEHPOUR, UCLA — When a drop is placed on a solid surface with temperature below solidification temperature, it comes to arrest at a finite time after deposition. This problem has a wide range of applications in engineering such as coating, ink jets and 3D printers. In the last 10 years, there have been very few studies of this paramount phenomenon. The physical parameters of the liquid and the substrate as well as spreading speed affect the arrest radius, the contact angle at arrest point, and time of arrest. Here, we use several different fluids to study these properties on the outcome of the spreading dynamics. The fluid is deposited using a syringe pump system to provide a constant flow rate for deposition. The evolution of wetting contact angle and base diameter of spreading drops were measured by a high speed digital camera. A parametric study of the radius of arrest and the arrested contact angle is provided.

8:52AM M4.00005 Drop dynamics on a thin film: Drop engulfment , PILNAM KIM, Princeton University, ANDREAS CARLSON, The Royal Institute of Technology, HOWARD A. STONE, Princeton University — When a liquid drop spreads on a thin film of another immiscible liquid, the liquid drop and film deform to minimize the surface energy. We investigate the dynamics of the motion of a water drop that comes in contact with a thin film of a silicone oil. We first present the engulfment of a water drop by a silicone film. We identify that the drop engulfment is dominated by the drop size and is independent of the thin film thickness. The interface where water/oil/air meets deforms as a spreading event, where the radius evolves as a power-law in time. If the solid that supports the thin oil film is hydrophobic, the film between the drop and the solid remains stable even if it is thinned by gravity, making the drop "float" on the solid. In the presence of a gradient in solid surface energy, the floating water drop moves toward the parts of the solid surface that have the highest energy. We suggest that this motion is caused by the imbalance in surface forces at the front and rear of the droplet, where the driving force is believed to originate from the interface-solid substrate interaction and acts through the thin silicone film.

9:05AM M4.00006 ABSTRACT WITHDRAWN —

9:18AM M4.00007 Motion and shape of partially non-wetting drops on inclined surfaces , BABURAJ A PUTHENVEETTIL, VIJAYA SENTHILKUMAR K, IIT Madras, E.J. HOPFINGER, LEGI-CNRS, IIT MADRAS- LEGI COLLABORATION — We study high Reynolds number (Re) motion of partially non- wetting liquid drops on inclined surfaces using (i) water on Fluoro-Alkyl Silane (FAS) coated glass and (ii) mercury on glass. The high hysteresis (35°) water drop experiments have been conducted for a range of inclination angles $26^\circ < \alpha < 62^\circ$ which give a range of Capillary numbers $0.0003 < Ca < 0.0075$ and $137 < Re < 3142$. For low hysteresis (6°) mercury on glass experiments, $5.5^\circ < \alpha < 14.3^\circ$ so that $0.0002 < Ca < 0.0023$ and $3037 < Re < 20069$. It is shown that when $Re \gg 10^3$ for water and $Re \gg 19$ for mercury, the observed velocities are accounted for by a boundary layer flow model. The dimensionless velocity in the inertial regime, $Ca\sqrt{Re}$ scales as the modified Bond number (Bo_m), while $Ca \sim Bo_m$ at low Re. We show that even at high Re, the dynamic contact angles (θ_d) depend only on Ca, similar to that in low Re drops. Only the model by Shikhmurzaev is consistent with the variation of dynamic contact angles in both mercury and water drops. We show that the corner transition at the rear of the mercury drop occurs at a finite, receding contact angle, which is predicted by a wedge flow model that we propose. For water drops, there is a direct transition to a rivulet from the oval shape at a critical ratio of receding to static contact angles.

9:31AM M4.00008 Buckling instability of a pinned droplet , GWYNN ELFRING, ERIC LAUGA, University of California San Diego — Pinned droplets have been experimentally observed to develop a shape instability when squeezed against a non-wetting surface. Using a combination of analysis and simulation we show that this instability occurs regardless of the contact angle of the surface due to a geometric bifurcation past a critical conformation in a manner reminiscent of a liquid bridge between two columns. We characterize the transition of a droplet from symmetric to asymmetric conformations and show how this leads to a buckling of the bubble under a load.

9:44AM M4.00009 Effects of particle number on interaction of particles with contact line in inkjet-printed evaporating colloidal drops , YING SUN, VIRAL CHHASATIA, Drexel University — The deposition behavior of inkjet-printed aqueous colloidal drops onto glass substrates with systematically varied wettability has been investigated by using fluorescence microscopy and a high-resolution goniometer. Real-time side-view images show that the contact angle of an evaporating colloidal drop is a function of the number of particles in suspension. The number of particles here is changed either by changing particle volume fraction while keeping particle size as a constant or by changing the particle size (10, 55, and 550 nm in radius) while keeping the particle volume fraction as a constant. During different stages of evaporation, the interplay of surface tension, drag due to evaporative flow, and particle-substrate interactions, rearranges particles inside a colloidal drop near the contact line region. These forces depend upon the size of the particles; however, the net effect of all these forces is independent of particle size. As the number of particles increase inside a drop, the receding contact angle of the evaporating drop decreases due to pinning of particles near the contact line. This reduction in receding contact angle increases the diameter of the particle deposition. The size of the particles affects the deposition diameter as smaller particles can move closer to the contact line compared to the larger particles and have a larger deposition diameter.

9:57AM M4.00010 Contact line pinning of a perfectly wetting and volatile liquid at a sharp edge[1] , YANNIS TSOUMPAS, SAM DEHAECK, ALEXEY REDNIKOV, PIERRE COLINET, Universite Libre de Bruxelles - TIPs, CP 165/67 — According to the Gibbs' criterion, a sharp edge can act as an energy barrier during the spreading of a liquid drop on a rigid substrate. In this study, however, we are trying to determine experimentally whether Gibbs' criterion is also valid for the case of a perfectly wetting and volatile liquid. To this purpose, we constructed a groove of triangular cross section on a Plexiglas substrate. During the experiments liquid was injected in the region surrounded by the groove, which had a square shape with rounded corners. The results indicated that a microgroove edge can indeed prevent the spreading of the liquid drop up to a certain extent, with the maximum apparent contact angle being in close agreement with the one given by Gibbs' criterion. Nevertheless, the apparent contact angle at breakup was found to be significantly lower in the corner of our region. To study this further we developed a static model, which takes into account surface tension and gravity but not the evaporation. Both the simulations and the experiments have confirmed a remarkable behaviour of the contact line at the corner. Finally, to grasp the effect of evaporation, experiments have also been conducted with drops of equally wetting but less volatile liquids.

[1]Supported by ESA & BELSPO, by the EU, and by FRS - FNRS.

Tuesday, November 22, 2011 8:00AM - 9:57AM —

Session M5 CFD VI: Combustion 308

8:00AM M5.00001 Turbulence and Combustion in Type Ia Supernovae[1] , AARON JACKSON[2], Stony Brook University, DEAN TOWNSLEY, The University of Alabama, ALAN CALDER, Stony Brook University — Turbulent combustion plays a critical role in Type Ia supernovae, bright astrophysical explosions that serve as cosmological distance indicators. The most successful scenario for reproducing observations involves a deflagration born in the turbulent core a massive C/O white dwarf that subsequently experiences a deflagration-to-detonation transition (DDT) due to turbulence-flame interaction (TFI), although unconfined DDT is poorly understood. Due to the highly non-linear nature of the explosion, the early flame propagation is critically important for determining the explosion outcome. We present full-star, 3D calculations of the deflagration phase of a SNIa explosion. As the flame evolves, it is subject to the Rayleigh-Taylor and Kelvin-Helmholtz instabilities as well as TFI. We analyze the resolved turbulence at the flame front throughout the evolution of the explosion and consider the necessity of modeling unresolved TFI. Furthermore, we consider whether conditions estimated for DDT are likely to occur given the turbulent intensity at the flame front.

[1] This work was supported by NASA under grant NNX09AD19G, the DOE under DE-FG02-87ER40317 and an INCITE allocation.

[2]Current address Naval Research Lab

8:13AM M5.00002 A Low-Dissipation Numerical Method for Compressible Multi-component and Reacting Flows , RYAN HOUIM, KENNETH KUO, The Pennsylvania State University — A low-dissipation for calculating multi-component gas dynamic flows with variable specific heat ratio that is capable of accurately simulating complex flows that contain both high- and low-Mach number features is presented. The numerical method combines features from the quasi-conservative double-flux multi-component model, high-resolution weighted essentially non-oscillatory schemes, and adaptive total variation diminishing slope limiters. Turbulence is implicitly modeled by using a low Mach-number adjustment procedure in conjunction with either AUSM or the HLLC approximate Riemann solver to evaluate numerical fluxes. To avoid spurious oscillations, characteristic variables are interpolated near shock waves and primitive variables are interpolated elsewhere for increased computational efficiency. Success of the method has been demonstrated with a series of numerical experiments including premixed deflagrations, Chapman-Jouget detonations, re-shocked Richtmyer-Meshkov instability, shock-wave and diffusion flame interactions, and multi-dimensional cellular detonations.

8:26AM M5.00003 An Irregularly Portioned FDF Solver , PATRICK PISCIUNERI, S. LEVENT YILMAZ, University of Pittsburgh, PETER STRAKEY, National Energy Technology Laboratory, PEYMAN GIVI, University of Pittsburgh — The "Irregularly Portioned Lagrangian Monte Carlo" (IPLMC) [1] in LES/FDF is extended to include the Eulerian flow solver in a coupled manner. The resulting methodology is for LES of reacting flows on massively parallel platforms, and is intended for LES of turbulent reacting flows described by complex kinetics. The new solver provides much improved scalability over its predecessor for utilization of a higher number of processors. Sample results are presented of LES of non-premixed flames, along with scalability benchmarks.

[1] Yilmaz, S. L., Nik, M. B., Sheikhi, M. R. H., Strakey, P. A., and Givi, P., An Irregularly Portioned Lagrangian Monte Carlo Method for Turbulent Flow Simulation, *J. Sci. Comput.*, **47**(1):109–125 (2011).

8:39AM M5.00004 Mesh quality metrics for large-eddy simulation of fire dynamics , RANDALL MCDERMOTT, NIST — The Fire Dynamics Simulator (FDS) is a large-eddy simulation code used in fire safety engineering. In this talk, we outline the implementation of three mesh quality metrics in FDS: (1) a measure of turbulence resolution based on a model for the fraction of unresolved kinetic energy (Pope, 2004), (2) a measure of scalar resolution based on a model for the unresolved scalar variance (Vervisch et al., 2010), and (3) a simple wavelet-based error measure. The metrics are examined in grid resolution studies of the McCaffrey fire plume experiments (McCaffrey, 1979), establishing target metric values for fire plume applications.

8:52AM M5.00005 A Parallel Adaptive Wavelet Method for the Simulation of Compressible Reacting Flows , ZACHARY ZIKOSKI, SAMUEL PAOLUCCI, University of Notre Dame — The Wavelet Adaptive Multiresolution Representation (WAMR) method provides a robust method for controlling spatial grid adaption — fine grid spacing in regions of a solution requiring high resolution (i.e. near steep gradients, singularities, or near- singularities) and using much coarser grid spacing where the solution is slowly varying. The sparse grids produced using the WAMR method exhibit very high compression ratios compared to uniform grids of equivalent resolution. Subsequently, a wide range of spatial scales often occurring in continuum physics models can be captured efficiently. Furthermore, the wavelet transform provides a direct measure of local error at each grid point, effectively producing automatically verified solutions. The algorithm is parallelized using an MPI-based domain decomposition approach suitable for a wide range of distributed-memory parallel architectures. The method is applied to the solution of the compressible, reactive Navier-Stokes equations and includes multi-component diffusive transport and chemical kinetics models. Results for the method's parallel performance are reported, and its effectiveness on several challenging compressible reacting flow problems is highlighted.

9:05AM M5.00006 Vortex Shedding of Various Bluff Bodies in a Cross Flow and Flame Luminosity , CHRISTOPHER RUSCHER, JOHN DANNENHOFFER, MARK GLAUSER, Syracuse University, BARRY KIEL, BALU SEKAR, Air Force Research Laboratory, ROBERT KAPAKU, Ohio State University — A comparison between flame luminosity and non-combusting flows was made in order to better understand the effects combustion and the flame have on the fluid dynamics of a flow. A large eddy simulation (LES) of a cylinder in a cross flow was done at different Reynolds number in order to make the comparison between flame luminosity and non-combusting flows using OVERSET grid technology. Proper orthogonal decomposition (POD) was done for the different cases in order to compare how much energy is contained in symmetric, asymmetric, and uncorrelated POD modes. Initial results show the modal energies of the non-combusting flow do not match the modal energies of the flame luminosity.

9:18AM M5.00007 Investigation of a subsonic round binary jet release , BORIS CHERNYAVSKY, PIERRE BÉNARD, Institut de Recherche sur l'hydrogène — We present results of an ongoing study of a binary H_2/CO jet/plume release. The original motivation behind this research lies in the need within the fuel cell community to establish safety criteria and the extent of potential danger envelope for unintended H_2/CO mixture releases for a variety of possible release conditions, including release velocity (jet versus plume type releases), orientation and release opening diameter, and initial gases fraction ratio, taking into account: a) the extent of flammable concentrations of H_2, corrected for the presence of CO; b) the extent of CO concentration exceeding health safety limits; c) the extent of flammable CO concentration. Presented are results of numerical simulations, covering a wide range of release conditions, including release velocity, orientation and gas fraction ratios. Particular attention has been paid to the gas segregation driven by a) buoyancy due to significant difference in gas densities; b) differences in diffusion properties, which could potentially affect jet evolution. It is shown that the observed segregation is small enough so that the concentration of one gas can be estimated from the measured concentration of another with sufficient degree of accuracy, except in the immediate vicinity of the origin of the jet. Subsequent simulations revealed that the primary driver of the segregation appears to be the difference in the diffusion properties between the mixture components.

9:31AM M5.00008 Algebraic decomposition as a seamless multiscale coupling strategy , NICOLAS HADJICONSTANTINOU, MIT — We discuss a new class of approaches for efficiently simulating multiscale kinetic problems, with particular emphasis on applications related to small-scale transport. These approaches, referred to as deviational, are based on a decomposition of the kinetic description into an equilibrium part, that is described deterministically (analytically or numerically), and the remainder, which is described using a particle simulation method. We show that it is possible to derive evolution equations for the two parts from the governing kinetic equation, leading to a decomposition that is dynamically and automatically adaptive, and a multiscale method that seamlessly bridges the two descriptions *without introducing any approximation.* Our discussion pays particular attention to stochastic simulation methods that are typically used to simulate kinetic phenomena; in this context, these decomposition approaches can be thought of as control- variate variance reduction formulations, with the nearby equilibrium serving as the control. Such formulations can provide substantial computational benefits in a broad spectrum of applications because a number of transport processes and phenomena of practical interest correspond to perturbations from nearby equilibrium distributions which can be exploited to drastically reduce the number of particles required in the simulation. In many cases the computational cost reduction is sufficiently large to enable otherwise impossible simulations.

9:44AM M5.00009 Modeling of heat transfer and mixing processes for complex geometries with applications to energy conversion systems[1] , NIKOLAOS PRASIANAKIS, JINFEN KANG, FELIX BUECHI, JOHN MANTZARAS, Paul Scherrer Institut, Switzerland — The study of advanced small energy conversion systems, such as fuel cells (SOFCs, PEFCs) and microcombustors, requires lattice Boltzmann models that can handle heat transfer and multicomponent species mixing. A new model is developed. The new thermal equilibrium populations are derived following the procedure described in Refs [1,2]. The collision step is split in two BGK-type relaxation processes as described in Ref. [3]. The resulting model has the capability of variable Prandtl and Schmidt numbers. The simplicity of the model allows its implementation to complex geometry flows. For an accurate prediction of the micro flow effects, the diffusive boundary condition is used. Micro-flow simulations of the flow through a SOFC are presented. Preliminary results of flows through three-dimensional complex geometries typically found in PEFCs are shown.

[1] N.I. Prasianakis, I.V. Karlin, PRE 76, 016702 (2007).
[2] N.I. Prasianakis, I.V. Karlin, J. Mantzaras, K. Boulouchos, PRE 79, 066702 (2009).
[3] S. Arcidiacono, I.V. Karlin, J. Mantzaras, C.E. Frouzakis, PRE 76, 046703 (2007).

[1]Supported by Bundesamt fur Energie (BFE) project 103078.

Tuesday, November 22, 2011 8:00AM - 10:10AM – Session M6 Turbulence Theory IV 309

8:00AM M6.00001 Linear flow dynamics near a T/NT interface[1] , MIGUEL TEIXEIRA, University of Lisbon, CARLOS SILVA, Technical University of Lisbon — The characteristics of a suddenly-inserted T/NT interface separating a homogeneous and isotropic shear-free turbulence region from a non-turbulent flow region are investigated using rapid distortion theory (RDT), taking full account of viscous effects. Profiles of the velocity variances, TKE, viscous dissipation rate, turbulence length scales, and pressure statistics are derived, showing very good agreement with DNS. The normalized inviscid flow statistics at the T/NT interface do not depend on the form of the assumed TKE spectrum. In the non-turbulent region, where the flow is irrotational (except within a thin viscous boundary layer), the dissipation rate decays as z^{-6}, where z is distance from the T/NT interface. The mean pressure exhibits a decrease towards the turbulence due to the associated velocity fluctuations, consistent with the generation of a mean entrainment velocity. The vorticity variance and dissipation rate display large maxima at the T/NT interface due to the existing inviscid discontinuities of the tangential velocity, and these maxima are quantitatively related to the thickness of the viscous boundary layer (VBL). At equilibrium, RDT suggests that the thickness of the T/NT interface scales on the Kolmogorov microscale.

[1]We acknowledge the financial support of FCT under Project PTDC/EME-MFE/099636/2008.

8:13AM M6.00002 PDF Equation Approach to the Lagrangian Velocity Increment Statistics of Fully Developed Turbulence , MICHAEL WILCZEK, Institute for Theoretical Physics, University of Muenster, HAITAO XU, Max Planck Institute for Dynamics and Self-Organization, NICHOLAS T. OUELLETTE, Department of Mechanical Engineering and Materials Science, Yale University, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization — The Lagrangian velocity increment statistics of fully developed three-dimensional turbulence is known to be highly intermittent, i.e., the probability density functions (PDFs) undergo a transition from a nearly Gaussian shape for large time lags to a highly non-Gaussian shape with pronounced tails for small time lags. We study this shape deformation across scale in the framework of the corresponding PDF equation, which is readily derived from the Lagrangian equations of motion. Here, the conditional acceleration plays a key role. The unclosed acceleration term is determined from experimental and direct numerical simulation data and is investigated with respect to the possibility of a simple mechanism for the origin of Lagrangian intermittency.

8:26AM M6.00003 The role of helicity on the turbulent energy flux , FEDERICO TOSCHI, Technische Universiteit Eindhoven (NL), LUCA BIFERALE, University of Tor Vergata (IT), STEFANO MUSACCHIO, CNRS, Laboratoire J.A. Dieudonne UMR 6621 (FR) — Three-dimensional turbulence is characterized by two inviscid constants of motion, kinetic energy and helicity. Helicity (i.e. the correlation between velocity and vorticity) is a key quantity in fluid turbulence, and has not a definite sign. This mere fact have since long been associated with the different phenomenology of 3d vs. 2d turbulence, which in contrast has two positive defined conserved quantities, energy and enstrophy (the square vorticity). In this study we investigate the role of helicity on the turbulent cascade process by decomposing the velocity field on the eigenmodes of the curl operator which have definite-sign helicity. We present numerical results on the statistical properties of the energy flux and velocity field in turbulent flows in which interactions between modes are selected according to their helicity. Minimal bibliography: P. D. Ditlevsen and P. Giuliani (2001) “Cascades in helical turbulence” PRE, 63(3), 036304. Q. Chen, S. Chen, and G.L. Eyink (2003) “The joint cascade of energy and helicity in three-dimensional turbulence” Physics of fluids, 15(2), 361. R. Benzi, L. Biferale, R. M. Kerr, and E. Trovatore (1996) “Helical shell models for three- dimensional turbulence” Phys. Rev. E 53, 3541. F. Waleffe (1992) “The nature of triad interactions in homogeneous turbulence” Phys. Fluids A, 4(2), 350.

8:39AM M6.00004 Lagrangian Acceleration Statistics in Curvilinear Coordinates , OLIVER KAMPS, Center for Nonlinear Science, University of Muenster, MICHAEL WILCZEK, Institute for Theoretical Physics, University of Muenster — In this talk we present results from the analysis of the Lagrangian acceleration statistics in curvilinear coordinates. The investigation is based on numerical simulations of isotropic, homogeneous turbulence in three dimensions for a certain range of Reynolds numbers. We focus on the effect of curvilinear coordinates on statistical observables like the cross correlation between acceleration components and the long time memory of the acceleration. Beside this we compare our observations to the corresponding observables in simulations of two-dimensional turbulence in the inverse cascade regime.

8:52AM M6.00005 Turbulence in an inhomogeneous medium , ROBERT RUBINSTEIN, NASA Langley Research Center — In some compressible turbulent flows, it is appropriate to assume that density fluctuations are negligible. The additional assumption that the mean density is known independently of the velocity field defines a model problem of turbulent fluctuations in an inhomogeneous medium in which the solenoidal condition on velocity of the constant density case is replaced by a solenoidal condition on the product of density and velocity. Because the fluctuations are inhomogeneous, the correlation function is not diagonal in the Fourier representation. Closure theories for this problem will be formulated and applied to analyze the possibility of inhomogeneous steady states and the applicability of gradient transport models. A generalized self-similarity for variable density will be developed and applied to the compressible law of the wall for boundary layers at moderate Mach numbers. Implications of the breakdown of this type of self-similarity as Mach number is increased will be drawn for understanding the behavior of cold-wall boundary layers.

9:05AM M6.00006 Flow-thermodynamics interactions in compressible shear-driven turbulence: Linear analysis of possible flow control strategies , REBECCA BERTSCH, GAURAV KUMAR, SHARATH GIRIMAJI, Texas A&M University — Flow-thermodynamics interaction in turbulent flows can be classified into three categories based on the action of pressure fluctuations. In very high Mach number flows, pressure fluctuations play an insignificant role as momentum far exceeds pressure forces. At very low Mach numbers, pressure is determined by the Poisson equation and flow-thermodynamics interactions are dynamically not very important. However, at intermediate Mach numbers, pressure exhibits wave character leading to critical flow-thermodynamics interactions and concurrent modification in the nature of turbulence. In our previous works, we have established that inhibiting influence of compressibility on turbulence is due to the intermediate Mach number regime. In this work, we use RDT (rapid distortion theory) linear equations to examine some strategies for flow control in the intermediate Mach number regime by exploiting flow-thermodynamic interactions. The results have important implications for inhibiting onset of turbulence in hypersonic external flows and intensifying mixing in internal propulsion flows.

9:18AM M6.00007 Statistics of incremental averages of passive scalar fluctuations[1] , COLIN MEYER, University of California Berkeley, Department of Civil and Environmental Engineering, LAURENT MYDLARSKI, McGill University, Department of Mechanical Engineering — Whereas statistics of differences in turbulent quantities measured over a given separation have been extensively studied, statistics of the incremental averages of the same quantities (e.g., $\Sigma\theta \equiv [\theta(x+r)+\theta(x)]/2$) have only been the focus of recent research. The present work studies incremental averages of a fluctuating passive scalar (temperature) in nearly homogeneous, isotropic turbulence, generated by an active grid, with an imposed mean scalar gradient. Following the arguments of Mouri and Hori (*Phys. Fluids*, 2010) for the velocity field, we derive a scale-dependent budget for the incremental average of the scalar field fluctuations, $\Sigma\theta$. We discuss its relationship to Yaglom's four-thirds law (which pertains to differences of passive scalar fluctuations) and compare the results with the experimental data. Furthermore, the statistics of $\Sigma\theta$ are compared with those of the incremental averages of the velocity fluctuations (Σu_α).

[1]Support provided by NSERC

9:31AM M6.00008 Nonlinear stochastic advection and structure generation in two-dimensional ideal fluids[1] , MADALINA VLAD, FLORIN SPINEANU, National Institute of Laser, Plasma and Radiation Physics — We show that trajectory trapping or eddying in the structure of the turbulence is the main physical reasons for the strong nonlinear effects that are observed in two-dimensional ideal fluids. This conclusion is drawn from a study of the statistics of test particles in turbulent Euler fluids, which is based on a new analytical method. Trajectory trapping determines non-Gaussian distribution of the displacements, quasi-coherent motion and local flows. This nonstandard statistical properties of trajectories determine the evolution of turbulence toward large correlation lengths (inverse cascade) and long correlation times. Thus, trajectory trapping explains the tendency of structure generation in two-dimensional Euler fluids.

[1]This work was supported by CNCSIS-UEFISCDI, project number PNII-IDEI 1104/2008

9:44AM M6.00009 Universal constants and equations of turbulent motion[1] , HELMUT BAUMERT, IAMARIS — For turbulence at high Reynolds number we present an analogy with the kinetic theory of gases, with dipoles made of vortex tubes as frictionless, incompressible but deformable quasi-particles. Their movements are governed by Helmholtz' elementary vortex rules applied locally. A contact interaction or "collision" leads either to random scatter of a trajectory or to the formation of two likewise rotating, fundamentally unstable whirls forming a dissipative patch slowly rotating around its center of mass, the latter almost at rest. This approach predicts von Karman's constant as 1/sqrt(2 pi) = 0.399 and the spatio-temporal dynamics of energy-containing time and length scales controlling turbulent mixing [Baumert 2005, 2009]. A link to turbulence spectra was missing so far. In the present contribution it is shown that the above image of dipole movements is compatible with Kolmogorov's spectra if dissipative patches, beginning as two likewise rotating eddies, evolve locally into a space-filling bearing in the sense of Herrmann [1990], i.e. into an "Apollonian gear." Its parts and pieces are are frictionless, excepting the dissipative scale of size zero. Our approach predicts the dimensionless pre-factor in the 3D Eulerian wavenumber spectrum (in terms of pi) as 1.8 , and in the Lagrangian frequency spectrum as the integer number 2. Our derivations are free of empirical relations and rest on geometry, methods from many-particle physics, and on elementary conservation laws only.

[1]Department of the Navy Grant, ONR Global

9:57AM M6.00010 Nonlinear Mechanisms in Streamwise Constant Plane Couette Flow , DENNICE F. GAYME, California Institute of Technology, BASSAM BAMIEH, University of California, Santa Barbara, BEVERLEY J. MCKEON, JOHN C. DOYLE, California Institute of Technology — The dominance of streamwise constant features was previously used to motivate input-output analysis of a streamwise constant (2D/3C) model of plane Couette flow (Gayme et. al 2011). Periodic spanwise-wall normal (z-y plane) stream functions acted as the input and led to a forced 2D/3C streamwise velocity field qualitatively similar to a fully turbulent spatial field of DNS data, in particular having a spanwise mean consistent with the time-averaged mean turbulent velocity profile. In this work, we further explore the relationship between the nonlinearity in the model and the mechanisms involved in creating the turbulent velocity profile's shape. We combine similar input functions with a weakly nonlinear (perturbation) analysis to determine analytical expressions for the leading terms in the streamwise velocity expansion. Our results provide further evidence of the importance of each of the nonlinear streamwise constant terms in the momentum transfer required to generate the turbulent mean velocity profile. **Acknowledgements:** This research is supported by AFOSR. B.J.M. gratefully acknowledges NSF-CAREER award no. 0747672 (program managers W. W. Schultz & H. H. Winter).

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M7 Turbulent Boundary Layers VII: High Re/Mach Nos. 310

8:00AM M7.00001 Well resolved pipe flow measurements at extreme Reynolds numbers[1] , MARGIT VALLIKIVI, MARCUS HULTMARK, Princeton University, SEAN BAILEY, University of Kentucky, ALEXANDER SMITS, Princeton University — Statistics of the streamwise velocity component were measured at Reynolds numbers ranging from $Re_\tau = 1\times10^3$ to $Re_\tau = 1\times10^5$ in fully-developed pipe flow. Pipes with two different surface finishes were studied, allowing a comparison between high Reynolds numbers turbulence in the hydraulically smooth and rough regimes. For better spatial resolution a nano- scale thermal anemometry probe (NSTAP) was used to acquire the data. The data gives new insight on pipe flow turbulence at extremely high Reynolds numbers, showing no interaction between the inner and the outer layer. The results also show a Reynolds number dependent outer peak in the turbulent fluctuations, which is only evident at very high Reynolds numbers. These extremely high Reynolds numbers reveal that the scaling of the fluctuations is much more similar to that of the mean velocities than previously believed.

[1]Supported under NR Grant N00014-09-1-0263 and NSF Grant CBET-1064257.

8:13AM M7.00002 Turbulence Spectra in High Reynolds Number Pipe Flow[1] , BRIAN ROSENBERG, MARCUS HULTMARK, MARGIT VALLIKIVI, ALEXANDER SMITS, Princeton University — Streamwise velocity spectra are acquired in fully-developed turbulent pipe flow over two decades of Reynolds number. Highly-resolved measurements are made possible by using a nano-scale thermal anemometry probe. We address the existence and nature of the classical k^{-1} and $k^{-5/3}$ spectral regions as well as the behavior of the very-large-scale motions (VLSM). The wall-normal variation of the VLSM spectral peak suggests that these motions originate at the boundary and grow with increasing wall distance, which stands in contrast to the outer-scaling behavior of boundary layer superstructures. This mechanism implies that in pipe flows outer- and inner- layer motions do not interact significantly, which was first seen in the behavior of the turbulence intensity.

[1]Supported under NR Grant N00014-09-1-0263 and NSF Grant CBET-1064257.

8:26AM M7.00003 Zero pressure gradient boundary layer at extreme Reynolds numbers[1] , MARCUS HULTMARK, MARGIT VALLIKIVI, ALEXANDER SMITS, Princeton University — Experiments were conducted in a zero pressure gradient flat plate boundary layer using the Princeton/ONR High Reynolds number Test Facility (HRTF). The HRTF uses highly compressed air, up to 220 atmospheres, to produce Reynolds numbers up to $Re_\theta = 225,000$. This corresponds to a $\delta^+ = 65,000$ which is one of the highest Reynolds numbers ever measured in a laboratory. When using pressure to achieve high Reynolds numbers the size of the measurement probes become critical, thus the need for very small sensors is acute. The streamwise component of velocity was investigated using a nanoscale thermal anemometer (NSTAP) as well as a $200\mu m$ pitot tube. The NSTAP has a spatial resolution as well as a temporal resolution one order of magnitude better than conventional measurement techniques. The data was compared to recent data from a high Reynolds number turbulent pipe flow and it was shown that the two flows are more similar than previous data suggests.

[1]Supported under NR Grant N00014-09-1-0263 (program manager Ron Joslin) and NSF Grant CBET-1064257(program manager Henning Winter).

8:39AM M7.00004 Turbulence in a Hypersonic Boundary Layer , OWEN WILLIAMS, ALEXANDER SMITS, Princeton University — Turbulent fluctuations in hypersonic boundary layers are conventionally collapsed using Morkovin scaling which has been shown to be broadly applicable up to Mach numbers as high as 5. To validate Morkovin's hypothesis at higher Mach numbers, and help improve our understanding of hypersonic wall-bounded turbulence, we report PIV measurements of two components of velocity fluctuations in a flat plate, turbulent boundary layer at Mach 7.4 in a perfect gas, at a Reynolds number based on momentum thickness of about 3500. Multiple tripping methods were evaluated to establish the sensitivity of the flow to initial conditions. Validation of the PIV results will be discussed and comparisons of various turbulent quantities will be made with DNS under identical flow conditions.

8:52AM M7.00005 Direct Numerical Simulation of Turbulent Boundary Layers on a Large Domains[1] , YIN-CHIU KAN, CLARA HELM, IZAAK BEEKMAN, PINO MARTIN, University of Maryland — Direct numerical simulation data of spatially-developing subsonic, supersonic and hypersonic turbulent boundary layers with matching similarity parameters are presented. The simulations are performed on large computational domains, using the rescaling technique with large rescaling lengths to minimize numerical correlation of the inflow and recycling planes. The simulations are run for long times without forcing the artificial acoustic modes in the free stream. We investigate the evolution of the boundary layer parameters and basic statistics with streamwise distance through the computational box and examine the turbulence structure through spectral analysis and filtered instantaneous flow fields. We pay special attention to the largest structures, with turbulence modeling, especially aspects unique to compressible boundary layers, in mind.

[1]This research is supported by AFOSR Grant Number AF 9550-09-1-0464 and the University of Maryland.

9:05AM M7.00006 DNS of a Mach 7.2 turbulent boundary layer over square bar roughness elements[1] , IZAAK BEEKMAN, PINO MARTIN, University of Maryland, CROCCO LAB TEAM — The direct numerical simulation of a turbulent, Mach 7.2 boundary layer over two-dimensional, square-bar roughness elements is presented. The physics of this flow are discussed, especially their similarities and differences with respect to compressible, smooth-wall, flat-plate boundary layers. Some interesting similarities exist, including the presence of near wall spanwise coherent structures and possible evidence of hairpin-packets. Additionally, the turbulence is characterized in terms of structure and statistics. The simulation technique and the challenges therein are also discussed, as the violent nature of the near wall region and the formation of transient shocks on the roughness elements makes this a challenging flow to study.

[1]This research is supported by NASA under Grant NNX08AD04A and the Air Force Office of Scientific Research under grant AF/9550-09-1-0464.

9:18AM M7.00007 Turbulence in supersonic boundary layers at moderate Reynolds number[1] , SERGIO PIROZZOLI, MATTEO BERNARDINI, Università di Roma "La Sapienza" — We study the organization of turbulence in supersonic boundary layers through direct numerical simulations at $M_\infty = 2$, $200 \leq Re_\tau \leq 1120$. Comparison of the velocity statistics up to fourth order shows collapse on reference incompressible data, provided the mean velocity and the velocity fluctuations are scaled to incorporate the effects of mean density variation, as postulated by Morkovin's hypothesis. Strict wall scaling is lost at high Reynolds for the "attached" flow variables, whose inner peak is found to increase with Re_τ. The modifications of the structure of the flow field that underlie this change of behavior are highlighted through flow visualizations, which substantiate the formation of large jet-like and wake-like motions in the outer part of the boundary layer, which extend their influence to the near-wall region. It is found that the typical size of the attached eddies roughly scales with the local mean velocity gradient, rather than being proportional to the wall distance, as dictated by the attached eddy hypothesis. The interactions of the large eddies in the outer layer with the near-wall region are quantified through the two-point amplitude modulation covariance, which characterizes the modulating action of energetic outer-layer eddies on other points in the same streamwise/wall-normal plane.

[1]The computational resources to perform the study were provided by CINECA.

9:31AM M7.00008 Spatial Correlation in the Fluctuating Wall Pressure Field Beneath a Supersonic Turbulent Boundary Layer , STEVEN BERESH, JOHN HENFLING, RUSSELL SPILLERS, BRIAN PRUETT, Sandia National Labs — Data have been acquired from a spanwise array of fluctuating wall pressure sensors beneath a wind tunnel wall boundary layer at Mach 2, then invoking Taylor's Hypothesis allows the temporal signals to be converted into a spatial map of the wall pressure field. Different frequency ranges of pressure fluctuations may be accessed by bandpass filtering the signals. This reveals signatures of coherent structures where negative pressure events are interspersed amongst positive events, with some degree of alternation in the streamwise direction. Within lower frequency ranges, streaks of instantaneously correlated pressure fluctuations elongated in the streamwise direction exhibit a spanwise meander and show apparent merging of pressure events, resembling similar structures known to exist in the velocity field. However, the pressure data lack the spanwise quasi-periodicity of positive and negative events found in velocity data, and conversely demonstrate a weak positive correlation in the spanwise direction whose extent increases at lower frequencies. The occasional passage of coherent structures spanning the entire sensor array provides an explanation for the weak spanwise correlations, a phenomenon not noted in velocity fields.

9:44AM M7.00009 Roughness Induced transition in supersonic boundary layers[1] , SUMAN MUPPIDI, KRISHNAN MAHESH, University of Minnesota — Laminar-to-turbulent transition at high speeds can significantly alter aerodynamic drag and heat transfer. The present study uses Direct Numerical Simulations to study transition of a Mach 2.9 boundary layer due to distributed surface roughness. Roughness causes the near-wall fluid to slow down and generates a strong shear layer over the roughness elements. Roughness surface also exerts an upward impulse on the fluid, generating counter-rotating pairs of streamwise vortices underneath the shear layer. As they move downstream, these vortices rise toward the shear layer, and their mutual interaction results in the break down of the shear layer, followed closely by transition to turbulence. The mean flow in the turbulent region shows a good agreement with available data for fully turbulent boundary layers. Simulations under varying conditions show that where the shear layer is not as strong, and the streamwise vortices are not as coherent, the flow remains laminar.

[1]This work is supported by NASA under the hypersonics NRA program grant NNX08AB33A.

9:57AM M7.00010 DSN at High Reynolds numbers of Thermal Turbulent Boundary Layers Subject to External Pressure Gradient[1] , LUCIANO CASTILLO, GUILLERMO ARAYA, Texas Tech University — Direct Numerical Simulations (DNS) of thermal spatially evolving boundary layers at zero (ZPG), favorable (FPG) and adverse (APG) pressure gradients are performed. The range for the momentum thickness Reynolds number, Re_θ, varies from 300 up to 3000. The predicted skin friction coefficient and Stanton number show fairly good agreement with empirical correlations, experimental and numerical data from the literature. The effects of the Reynolds number and streamwise adverse pressure gradient on the flow and thermal parameters are also explored and visualized. Additionally, the location of the maximum thermal fluctuations in outer coordinates moves closer to the wall as the Reynolds number increases. It was confirmed that the principal effects of adverse pressure gradient on the flow were the appearance of peaks, particularly on the streamwise velocity fluctuations, in the outer region. Furthermore, as the Reynolds number increases, the location of the maximum thermal fluctuations moved closer to the wall when plotted in outer coordinates. Finally, the observed correlation between streamwise velocity and thermal fluctuations in ZPG flows disappears in Strong APG flows.

[1]Funded by the Office of Naval Research.

Tuesday, November 22, 2011 8:00AM - 10:10AM – Session M8 Non-Newtonian Flows II 311

8:00AM M8.00001 Re-examing the logarithmic dependence of the mean velocity distribution in polymer drag reduced wall-bounded flow , CHRISTOPHER WHITE, University of New Hampshire, YVES DUBIEF, University of Vermont, JOE KLEWICKI[1], University of New Hampshire — The logarithmic dependence of the mean velocity distribution in polymer drag reduced wall-bounded flows is re-examined to study the effect of drag-reducing polymers on the von Kármán constant and to determine if the "ultimate profile," corresponding to the state of maximum drag reduction, is truly logarithmic. The results of our findings show very different behaviors of the mean velocity distribution in polymer drag reduced flows than the classical view. First it is shown that at low drag reduction (DR) the slope of the logarithmic overlap region increases with increasing DR. Next, at some relatively high DR (which likely depends on Re), the inertially dominated logarithmic overlap region is eradicated. After which, the mean velocity distribution has a similar shape as a laminar flow and the perceived logarithmic scaling behavior corresponds to a thickened buffer layer and is not truly logarithmic.

[1]Also at University of Melbourne

8:13AM M8.00002 Elastic turbulence in high Reynolds number polymer drag reduced flows , YVES DUBIEF, School of Engineering, University of Vermont, Burlington VT, CHRISTOPHER WHITE, Dept. of Mechanical Engineering, University of New Hampshire, Durham NH — The present study discusses the existence of small scale dynamics resembling elastic turbulence in polymeric transitional and maximum drag reduction (MDR) flows. The observed flow patterns are driven by elastic stress and occur in regions of very low turbulence found before and after the breakdown of nonlinear instabilities in polymeric transitional flows leading to MDR. A state of polymer-dominated spanwise instabilities was found, resulting in a structure of the wall shear quite different than the structures observed in transitional Newtonian flow. Similar instabilities are observed in the wake of the head of hairpin vortices in simulated MDR flows, an extended region of extensional flow of the order of the Kolmogorov scale in the normal direction . The important Reynolds number is not that of flow ($Re_\tau = 300$ and 600 for the Newtonian flows) but that of the local turbulent flow, which according to Kolmogorov approaches unity in the above mentioned flows, a reasonable magnitude for elastic turbulence. The existence of small scale elastic turbulence in transitional and MDR flows explains the phenomenon of early turbulence first observed in the 70s and challenges the notion that, in drag reduced flows, the energy flows only from large to small scales and never goes back from polymers to flow.

8:26AM M8.00003 Elastic turbulence in Taylor-Couette Flow of Dilute Polymeric Solutions: A Direct Numerical Simulation Study[1] , NANSHENG LIU, BAMIN KHOMAMI, Department of Chemical and Biomolecular Engineering University of Tennessee, Knoxville — Despite tremendous progress in development of numerical techniques and constitutive theories for polymeric fluids in the past decade, Direct Numerical Simulation (DNS) of elastic turbulence has posed tremendous challenges to researchers engaged in developing first principles models and simulations that can accurately and robustly predict the dynamical behavior of polymeric flows. In this presentation, we report the first DNS of elastic turbulence in the Taylor-Couette (TC) flow. Specifically, our computations with prototypical constitutive equations for dilute polymeric solutions, such as the FENE-P model are capable of reproducing the essential features of the experimentally observed elastic turbulence in TC flow of this class of fluids, namely, randomly fluctuating fluid motion excited in a broad range of spatial and temporal scales, and a significant increase of the flow resistance. Moreover, the experimentally measured Power Spectral Density of radial velocity fluctuations, i.e., two contiguous regions of power-law decay, -1.1 at lower frequencies and -2.2 at high-frequencies is accurately computed.

[1]We would like to thank NSF through grant CBET-0755269 and NSFC through grant NO. 10972211 for supporting of this work.

8:39AM M8.00004 Turbulence structure and statistics in polymer-induced low vs. high drag reduction regimes: the origin of the maximum drag reduction asymptote[1]

KYOUNGYOUN KIM, Hanbat National University, RADHAKRISHNA SURESHKUMAR, Syracuse University — Turbulent statistics and structure in polymer-induced low (20%) and high (66%) drag reduction (DR) regimes are studied via channel flow DNS at Re_τ =395. The initial hairpin eddy extracted from the conditional averages of the Q2 events is self-consistently evolved in the presence of polymer stresses using the FENE-P model. For low DR (Weissenberg number We_τ defined as the ratio of the polymer relaxation time λ to the viscous time scale = 50), large counter polymer torque is observed only near the vortex legs. This suppresses the auto-generation of new vortices primarily only in the inner layer. For high DR (We_τ ≥100), large counter polymer torque appears near the hairpin head and legs. This modifies both the inner and outer layer dynamics. When the Elasticity number E, defined as the ratio of λ to the eddy turnover time, approaches unity, the effect of the polymer encompasses the whole channel and DR approaches an asymptotic value. This distinction between low and high DR dynamics is not robustly captured by low Re simulations in which E remains $O(1)$.

[1]Supported by NSF Grant CBET 1055219 and by NRF Grant 2010-0007901.

8:52AM M8.00005 The maximum drag reduction asymptote

BJÖRN HOF, DEVRANJAN SAMANTA, Max Planck Institute for Dynamics and Self Organization, CHRISTIAN WAGNER, Saarland University — Addition of a small amount of long chain polymers to a Newtonian solvent can lead to a dramatic drag reduction in turbulent flows. This effect has been extensively studied since its discovery in the late 1940's. The drag reduction at first is proportional to the polymer concentration (Weisenberg number) but then saturates to the maximum drag reduction (MDR) asymptote. It is commonly believed that drag reduction results from an adjustment of the turbulent flow structure due to the action of the polymers. We here present experimental results of turbulent pipe flows using dilute polyacrylamid solutions at relatively large Weisenberg numbers (∼10). Our results show that for relatively low polymer concentrations transition to turbulence is postponed to higher Reynolds numbers. However when the Weisenberg number is increased further we find that the subcritical transition to turbulence, typical for Newtonian pipe flow disappears. Instead a supercritical instability is found at much lower Reynolds numbers which gives rise to a disordered flow. The observed drag of this disordered flow is identical to the well known MDR asymptote.

9:05AM M8.00006 Grid Turbulence in PEO solutions[1]

PETER MONKEWITZ, RICHARD VONLANTHEN, Swiss Federal Institute of Technology Lausanne (EPFL) — Grid turbulence in dilute PEO solutions is studied experimentally in a small, closed loop hydraulic tunnel. To attain higher Reynolds numbers based on the Taylor microscale of the order of 100 and an inertial range of about one decade, a novel passive grid with tethered spheres has been developed. By carefully studying the evolution of turbulence spectra as function of the age of the PEO solution, i.e. of the degradation of polymer molecules, it has been possible to clearly identify a time-dependent "Lumley" wave number κ_L where the fluid behavior switches abruptly from Newtonian to visco-elastic. This switch is characterized by a rather sharp transition from the Kolmogorov $\kappa^{-5/3}$ slope of the energy spectrum to a κ^{-3} slope. Dimensional analysis shows that this corresponds to a switch from constant down-scale energy flux to a self-regulated constant eddy rate of strain. A simple model is proposed for the time-dependence of the Lumley scale κ_L.

[1]the support of the Swiss NSF is gratefully acknowledged

9:18AM M8.00007 Draw resonance in non-isothermal non-Newtonian viscous sheets

ZHEMING ZHENG, CHUNFENG ZHOU, OLUS BORATAV, Corning Incorporated — In this presentation, we will discuss the instability known in literature as the "draw resonance" for a non-Newtonian viscous sheet of glass with non-isothermal conditions. Both eigen-solutions and transient solutions are used in the stability analysis. We will focus our discussion on the effects of viscoelasticity and thermal conditions (local or global variations) on the draw resonance stability. Our study found that the stability can be enhanced by both viscoelastic effect and thermal heating. It also demonstrated that the critical draw ratio is increasing significantly with the Deborah number and is sensitive to how the sheet is heated or cooled. We will also present stability results comparing sheet draw and fiber draw.

9:31AM M8.00008 Nonlinear elastic instabilities in parallel shear flows

LICHAO PAN, University of Pennsylvania, ALEXANDER MOROZOV, University of Edinburgh, PAULO ARRATIA, University of Pennsylvania — It is a common assumption that, in the absence of inertia and curvature, the flow of a viscoelastic fluid is linearly stable to flow perturbations. Recent evidence, however, suggests that such flow may be unstable to a finite amplitude perturbation. In this talk, we present evidence of a subcritical nonlinear instability for the flow of a dilute polymeric solution in a straight microchannel (no curvature) at low *Re*. The experimental configuration consists of a long, straight microchannel that is 100 μm deep, 100 μm wide and 3.0 cm long. The channel is divided into two main regions: a short (∼0.3 cm) region where an array of cylinders is positioned in order to introduce perturbations in the flow, and a long (∼2.7 cm) parallel flow region; a channel devoid of cylinders is also used for control. The flow is investigated using both dye advection and particle tracking velocimetry. Results show large velocity fluctuations far downstream (2 cm) away from the initial perturbation for strong enough and long lived disturbances. Small disturbances decay quickly under the same flow conditions (i.e. flow rate). A hysteresis loop, characteristic of subcritical instabilities, is observed.

9:44AM M8.00009 Visco-plastic Lubrication: New Areas for Application

SARAH HORMOZI, IAN FRIGAARD, Department of Mechanical Engineering and Mathematics, University of British Columbia — Stable multi-layer flows can be achieved at high Reynolds numbers by using a yield stress fluids in a lubricating outer layer. These flows have been demonstrated to be linearly and nonlinearly stable as well as observable experimentally; see Frigaard (2001), Moyers-Gonzalez et al. (2004) and Huen et al. (2007). Recently, we have studied these flows computationally in the setting of a Newtonian core fluid surrounded by a Bingham lubricated fluid, within pipe and channel configurations; see Hormozi et al. (2011a) and Hormozi et al. (2011b). The results show that we are able to freeze in non-planar interface and form interesting patterns by retaining an unyielded plug region at the interface. Our studies open up new potential areas for application such as drop encapsulation and near net shape production of multi-layered products with axial variations. We give an overview of experimental results on establishing these exotic patterns.

References: I.A. Frigaard, J. Non-Newt. Fluid Mech., 100, (2001) 4976. M. Moyers-Gonzalez, I.A. Frigaard & C. Nouar, J. Fluid Mech., 506, (2004) 117146. C.K. Huen, I.A. Frigaard & D.M. Martinez, J. Non-Newt. Fluid Mech., 142, (2007) 150161. S. Hormozi, K. Wielage-Burchard & I.A. Frigaard, J. Fluid Mech.,673, (2011) 432 467. S. Hormozi, K. Wielage-Burchard & I.A. Frigaard, J. Non-Newt. Fluid Mech.,166, (2011) 262278.

9:57AM M8.00010 Dynamics of inertialess flows of viscoelastic fluids: the role of uncertainty

MIHAILO JOVANOVIC, BINH LIEU, SATISH KUMAR, University of Minnesota — We use techniques from control theory to demonstrate high sensitivity of inertialess channel flows of viscoelastic fluids. To counter this sensitivity we explicitly account for modeling imperfections, such as the approximate nature of polymer constitutive equations, by quantifying their influence on transient and asymptotic dynamics. Our approach has strong connections to the analysis of pseudospectra of linear operators, and it exhibits the importance of streamwise-elongated flow patterns in viscoelastic fluids. For streamwise-independent flows with high elasticity numbers and finite Weissenberg numbers, We, we establish that the energy of velocity and polymer stress fluctuations scale as $O(We^2)$ and $O(We^4)$, respectively. This suggests that small amount of modeling uncertainty can destabilize nominally stable dynamics and promote transition to elastic turbulence. The underlying physical mechanism involves polymer stretching that introduces a lift-up of flow fluctuations similar to vortex tilting in inertia-dominated flows. The phenomenon examined here provides a possible route for the early stages of a bypass transition to elastic turbulence and might be exploited to enhance mixing in microfluidic devices.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M10 General Fluid Dynamics II 313

8:00AM M10.00001 Numerical simulation of a plunging flexible hydrofoil and its experimental validation[1] , TAO YANG, LEONARDO MARTIN-ALARCON, MINGJUN WEI, FANGJUN SHU, New Mexico State University — A monolithic approach for simulation of flexible flapping wings in fully-coupled motion has recently been developed. The methodology is based on a uniform description of fluid and structure in Eulerian framework. Immersed boundary technique is used to represent solid stress, solid-fluid interface, and active flapping motion in an overall Cartesian coordinate. In the current presentation, the focus is to apply the method on a simple two-dimensional problem of plunging flexible hydrofoil and then compare to the experimental results for validation. The three-dimensional results and experimental validations will also be discussed.

[1]Supported by Army High Performance Computing Research Center

8:13AM M10.00002 Experimental study of flow field around a plunging flexible hydrofoil[1] , LEONARDO MARTIN-ALARCON, TAO YANG, FANGJUN SHU, MINGJUN WEI, New Mexico State University — Recent developments in micro air vehicles (MAVs) have led to the improvement of computational fluid dynamics (CFD) simulations capable of simulating flexible flapping wing phenomena. For validation of these simulations, an experimental methodology is applied to characterize the flow physics involved with an immersed flexible flapping hydrofoil. Using a one-degree of freedom crank-shaft system, a silicone hydrofoil was actuated to flap under various kinematic conditions. The hydrofoil was subject to active plunging and passive pitching motion in both water and aqueous glycerin solutions. Phase-locked particle image velocimetry (PIV) measurements were obtained around the flapping hydrofoil. These measurements, along with force measurements using a six-axis load cell, are used to compare the results with those of the numerical simulations. By comparing the hydrofoil deformation, vortex evolution and force generation, good agreements between CFD and experimental results were observed.

[1]Supported by Army High Performance Computing Research Center

8:26AM M10.00003 The analysis of three dimensional flow around a low-aspect-ratio control fin with end plate , CHULMIN JUNG, KURNCHUL LEE, CHANKI KIM, Agency for Defense Development — Flow around the tip of control fin is fully three dimensional, and the prediction of lift on the control fin is essential in maneuvering a vehicle in an appropriate manner. Three dimensional flow effect on a low- aspect-ratio control fin is more obvious than on a moderate or high-aspect-ratio control fin. The three dimensional flow effect can be reduced by applying end plate. Through numerical simulations, we examine the flow field around a low-aspect-ratio control fin with and without end plate for different flap angles. The pressure, vorticity, lift and torque on the control fin are analyzed and compared to experiments.

8:39AM M10.00004 Effect of gust on flow patterns around a robotic hummingbird wing[1] , ELOY N.M. GONZALEZ, New Mexico State University, B.J. BALAKUMAR, Los Alamos National Laboratory, FANGJUN SHU, New Mexico State University — Several studies have demonstrated the importance of the leading edge vortex in enhancing lift production during hovering flight for a hummingbird. Most of these studies were performed under steady inflow conditions. However, real-life ornithopters in the field have to routinely tackle directional changes in the wind and gust. In this work, we investigate flow field variations around a hummingbird wing under well-controlled gust conditions using a 2-degree of freedom robotic model in a water channel. Conditions of gust are simulated by mounting the model on a translation stage, which allows the control of gust profiles. Phase-locked and time-resolved particle image velocimetry (PIV) measurements were obtained around the wing in the presence of various gust levels. These measurements, in combination with force and moment measurements using a six-axis load cell, are used to understand transient flow phenomena induced by the gust, and their effect on the net thrust and lift forces over a range of Reynolds number ($2100<Re<30000$).

[1]Supported by Continuum Dynamics Inc.STTR grant PO06-930 and Army High Performance Computing Research Center.

8:52AM M10.00005 Drag on Flexible, Slender Bodies Streamlined in Turbulent Flow , JEFFREY ROMINGER, HEIDI NEPF, Department of Civil & Environmental Engineering, Massachusetts Institute of Technology — Dynamic reconfiguration in response to turbulent fluid forcing alters the instantaneous drag forces on flexible, slender bodies, such a submerged aquatic plants. In this study we experimentally investigate the role of plant morphology, specifically length and structural stiffness, in reducing instantaneous drag forces on flexible plastic strips streamlined in turbulent flow. We limit this investigation to the stable regime of fluid- solid interactions, which is typical of many species of submerged aquatic vegetation. First the structural stiffness of the flexible strips is varied to determine how bending in response to turbulent fluid forces can alter instantaneous drag forces. We also investigate the role played by the ratio of the turbulence lengthscale to the length of the plastic strips. When the turbulence lengthscale is similar to or larger than the strip length, turbulent forces can act coherently over the length of the strip, which may enhance instantaneous forces. In contrast, when the turbulence lengthscale is smaller than the strip, the instantaneous forces can be non-coherent and interfere destructively, reducing the effect of the instantaneous forces on the strip.

9:05AM M10.00006 Flow-induced oscillations of a pipe conveying fluid with base excitation , GARY CHANG, YAHYA MODARRES-SADEGHI — We study various dynamic instabilities in a cantilevered pipe conveying fluid with base excitation. A plain cantilevered pipe conveying fluid loses its stability by a Hopf bifurcation, leading to either planar or non-planar flutter for flow velocities beyond the critical flow velocity for the Hopf bifurcation. If a mass is attached to the end of the pipe (a so-called added mass), the resulting dynamics becomes much richer, showing 2-D and 3-D quasi-periodic and chaotic oscillations at high flow velocities. In this work, we consider a cantilevered pipe subjected to either a periodic base excitation, or a periodic point excitation at a given location along the pipe, and study the response of the fluid-conveying pipe in 2-D and 3-D. It turns out that by selecting the right frequency and amplitude for the external excitation, we can force the non-planar oscillations to planar ones.

9:18AM M10.00007 The Hydrodynamic Optimization of a Cable-Flying Vehicle , BRIAN AMARAL, University of Rhode Island — Profiling the water column with high vertical and horizontal resolution in the deep ocean from a moving vessel is difficult with the current tools. Thus, a new, higher resolution-capable wire-flying vehicle, the WiFly, is being designed to profile the water column. The WiFly traverses up and down a ship-towed cable by generating lift from wings that protrude from the sides of vehicle. Because the success of the vehicle is based on its glide slope performance, the ratio of vertical velocity to horizontal velocity, the shape of the vehicle must be optimized for the flow conditions associated with its anticipated uses in oceanographic science. Results include evaluation methods, body shapes, and the parameters involved in selecting the most desirable shape characteristics.

9:31AM M10.00008 Physics of badminton shuttlecocks. Part 1 : aerodynamics , CAROLINE COHEN, BAPTISTE DARBOIS TEXIER, DAVID QUÉRÉ, CHRISTOPHE CLANET, LadHyX, Ecole Polytechnique — We study experimentally shuttlecocks dynamics. In this part we show that shuttlecock trajectory is highly different from classical parabola. When one takes into account the aerodynamic drag, the flight of the shuttlecock quickly curves downwards and almost reaches a vertical asymptote. We solve the equation of motion with gravity and drag at high Reynolds number and find an analytical expression of the reach. At high velocity, this reach does not depend on velocity anymore. Even if you develop your muscles you will not manage to launch the shuttlecock very far because of the "aerodynamic wall." As a consequence you can predict the length of the field. We then discuss the extend of the aerodynamic wall to other projectiles like sports balls and its importance.

9:44AM M10.00009 Analysis of the Free Rising Sphere in Newtonian Fluid and the "Pop Off" Effect[1] , ANNA SHCHETININA, JULIO GARCIA, EMMANUEL GAILLARD, LYES KADEM, HOI DICK NG, Concordia University, Montreal, Canada — In this experiment motion of a free rising sphere in Newtonian fluid is studied. Spheres of various densities and diameters are placed at the bottom of a large water tank and then released with zero initial velocity and no distortion in trajectory. The motion of the rising sphere is described in three phases: ascending, where the body undergoes constant acceleration and moves vertically while developing vortices on its both sides; transitional, where a secondary sideway motion is added and vortex shedding begins; and, finally, oscillatory, where the sphere begins to move in a sinusoidal pattern while describing even spiral trajectory. The mode in which the ball exits the water varies depending on several conditions. Depending on the Reynolds number, it escapes vertically or diagonally, which is referred to by Bourrier et al. (1984) [Eur. J. Phys. 5:225-231] as the 'pop off' effect. Using Particle Image Velocimetry (PIV), high-speed photography and image processing techniques, we discuss the causes of the oscillatory motion as well as the "pop off" effect produced by a sphere under placed conditions.

[1]This work is supported by the Natural Sciences and Engineering Research Council of Canada (NSERC) and the Canada Foundation for Innovation (CFI).

9:57AM M10.00010 Tubular jet generation by means of a pressure pulse induced by an eddy-current actuator , ALEXANDER KLEIN, Robert Bosch GmbH, IVO R. PETERS, GERBEN MORSINK, CHAO SUN, DEVARAJ VAN DER MEER, University of Twente, ROBERT GIEZENDANNER-THOBEN, Robert Bosch GmbH, DETLEF LOHSE, University of Twente — The generation and the evolution of tubular jets is studied experimentally and compared to numerical results from a boundary integral code. The jets are created at the free surface above a liquid column of purified water. An eddy-current actuator driven by a high voltage capacitor bank is used to create a pressure pulse with a duration of about $100\ \mu$s and varying amplitude of up to 70 bar. The pressure pulse travels in the vertical water column of length 1 m before hitting the free surface in a capillary tube of $4-8$ mm in diameter. The process of jet formation is captured using high-speed imaging at up to 60 kHz, while the pressure pulse is recorded by two PVDF transducers at 20 MHz. The recordings and the numerical simulations enable us to study the effect of the control parameters on the jet velocity (which can reach up to 50 m/s) and the mass flow. Namely, we study the effect of the applied acoustic power of the pressure pulse and the initial curvature of the free surface.

Tuesday, November 22, 2011 8:00AM - 10:10AM – Session M11 Surface Tension III 314

8:00AM M11.00001 Autophobing on a liquid substrate[1] , ELLEN PETERSON, STEPHEN GAROFF, ROOMI KALITA, RAMANKUR SHARMA, Carnegie Mellon University — When a drop of fluid is deposited on another fluid it may either completely spread over the underlying fluid or it may reach an equilibrium shape, assuming the two fluids are immiscible. The choice of end state is predicted by the sign of the spreading coefficient ($S = \Sigma^F - \Sigma^D - \Sigma^{DF}$) which relates the surface tension of the three interfaces: fluid/air (Σ^F), drop/air (Σ^D), and drop/fluid (Σ^{DF}). We experimentally investigate this behavior but discover that a static lens may form even when the spreading coefficient predicts complete spreading. We measure the surface tension of the underlying fluid and deduce that fluid escapes the contact line of the drop. If we allow the surface tension of the subphase to change due to this escaping fluid, we find the resulting spreading coefficient predicts a static lens. We compare the results to a mathematical model and confirm the observed spread area. The results of this investigation suggest that the lens resists flowing over the escaped layer of the same fluid, the mechanism of autophobing.

[1]NSF CBET-0931057, NSF DMS-0635983

8:13AM M11.00002 In situ, noninvasive characterization of superhydrophobic coatings[1] , G.C. TEPPER, M.A. SAMAHA, H. VAHEDI TAFRESHI, M. GAD-EL-HAK, Virginia Commonwealth University — Light scattering was used to measure the time-dependent loss of air entrapped within a submerged microporous hydrophobic surface subjected to different environmental conditions. The loss of trapped air resulted in a measurable decrease in surface reflectivity and the kinetics of the process was determined in real time and compared to surface properties, such as porosity and morphology. The light-scattering results were compared with measurements of skin-friction drag, static contact angle, and contact-angle hysteresis. The *In situ*, noninvasive optical technique was shown to correlate well with the more conventional methods for quantifying surface hydrophobicity, such as flow slip and contact angle. *In situ* characterization of submerged hydrophobic surfaces using light scattering represents a new and useful tool for real-time estimation of hydrophobicity and drag reduction.

[1]Financial support from the Defense Advanced Research Projects Agency (DARPA), contract number W91CRB-10-1-0003, is acknowledged.

8:26AM M11.00003 Salinity Effects on Superhydrophobic Coatings[1] , F.O. OCHANDA, M.A. SAMAHA, H. VAHEDI TAFRESHI, G.C. TEPPER, M. GAD-EL-HAK, Virginia Commonwealth University — Experiments are carried out to investigate the effect of NaCl concentrations on degree of hydrophobicity and longevity of polystyrene fibrous coating. A rheological study using salt water as a test fluid is performed to observe the generated drag reduction from the coating with increasing salt concentration compared to deionized water. Contact-angle measurements of droplets of solutions on the surface are used to validate the results from the rheometer. In situ noninvasive optical spectroscopy system is used to measure the time-dependent loss of entrapped air within the submerged fibrous coating. water for comparison. The superhydrophobic coating used is made of polystyrene fibers that are deposited using DC-biased AC-electrospinning. Such fabrication methods are far less expensive than ordered-microstructured fabrications, bringing the technology closer to large-scale submerged bodies such as submarines and ships. The present study sheds some light on how properties of a superhydrophobic coating could be influenced by water salinity.

[1]Financial support from DARPA, contract number W91CRB-10-1-0003, is acknowledged.

8:39AM M11.00004 Modeling superhydrophobic surfaces comprised of random roughness[1]

M.A. SAMAHA, H. VAHEDI TAFRESHI, M. GAD-EL-HAK, Virginia Commonwealth University — We model the performance of superhydrophobic surfaces comprised of randomly distributed roughness that resembles natural surfaces, or those produced via random deposition of hydrophobic particles. Such a fabrication method is far less expensive than ordered-microstructured fabrication. The present numerical simulations are aimed at improving our understanding of the drag reduction effect and the stability of the air–water interface in terms of the microstructure parameters. For comparison and validation, we have also simulated the flow over superhydrophobic surfaces made up of aligned or staggered microposts for channel flows as well as streamwise or spanwise ridge configurations for pipe flows. The present results are compared with other theoretical and experimental studies. The numerical simulations indicate that the random distribution of surface roughness has a favorable effect on drag reduction, as long as the gas fraction is kept the same. The stability of the meniscus, however, is strongly influenced by the average spacing between the roughness peaks, which needs to be carefully examined before a surface can be recommended for fabrication.

[1]Financial support from DARPA, contract number W91CRB-10-1-0003, is acknowledged.

8:52AM M11.00005 Spreading of thin rotating films: Competition of thermal Marangoni, centrifugal, and gravitational forcing

JOSHUA DIJKSMAN, Duke University, SHOMEEK MUKHOPADHYAY, UC Riverside, ROBERT BEHRINGER, Duke University — We experimentally probe the quasi static spreading of a sessile drop on a substrate under the influence of competing gravitational, thermo capillary (Marangoni) and centrifugal forcing. We use silicone oil on a prewetted silicon wafer, and we employ an interferometric technique to study the evolution of the film height profile. We discuss our results in the context of recent theory developed by Bostwick et al.

9:05AM M11.00006 On integrable singularities and apparent contact angles within a classical paradigm: partial and complete wetting regimes with or without phase change[1]

PIERRE COLINET, ALEXEY REDNIKOV, Universite Libre de Bruxelles - TIPs (Transfers, Interfaces and Processes), CP 165/67 — Far from claiming any ultimate resolution of the contact line paradoxes, we argue that a somewhat controversial paradigm, originally employed by de Gennes and collaborators, actually appears both to be quite reasonable at its foundations and to lead to physically consistent final results in a wide variety of situations. Curiously enough, while containing a singularity in itself, the approach nonetheless renders the classical contact-line singularities – both hydrodynamic and thermal – integrable, in particular as far as several quantities of interest are concerned. It is also readily applicable to quite a few situations: from equilibrium shapes and moving contact lines of a non-volatile liquid, to cases with evaporation into either a pure-vapor or an inert-gas atmosphere. The paradigm actually consists in an approach involving both the (positive or negative) spreading coefficient and the disjoining pressure in the form of a positive inverse cubic law, a conceptual framework that most notably describes structures with truncated precursor films on a macroscopically bare solid surface.

[1]Supported by ESA & BELSPO, by the EU - MULTIFLOW project, and by FRS - FNRS.

9:18AM M11.00007 On possible resolutions of the micro-edge problem for a contact line moving on a solid in a pure-vapor atmosphere with no slip and without extended precursor films[1]

ALEXEY REDNIKOV, PIERRE COLINET, ULB-TIPs, CP 165/67 — At the previous APS/DFD meeting, we reported on the possibility of a singularity-free description of a moving contact line totally in the framework of classical physics (no disjoining pressure, no slip) thanks to the phase change whose intensity is regulated by Kelvin effect. Pushing this idea further, there remains a related more practical issue: the Kelvin effect, even though classical, becomes apparent at such small scales that the disjoining pressure (DP) may nonetheless play a significant role. Here we show, for a class of DP isotherms remaining finite at zero film thicknesses, that the same kind of natural singularity-resolution mechanism still holds. As before, we consider, in the framework of the lubrication theory and a classical one-sided model, a contact line moving at a constant velocity (advancing or receding) and starting abruptly at a (formally) bare solid surface, the micro- contact angle being either equal to zero or finite. Also touched upon is a possible resolution of the remaining integrable singularities within the paradigm of de Gennes and coworkers, considered here in another talk by the present authors, by means of regularizing the DP isotherm at very small thicknesses.

[1]Supported by ESA & BELSPO, by the EU, and by FRS - FNRS.

9:31AM M11.00008 Impregnation transition in a powder[1]

PASCAL RAUX, HELOISE COCKENPOT, DAVID QUERE, CHRISTOPHE CLANET, Physique et mécanique des milieux heterogenes, CNRS, ESPCI, Paris, France & Ladhyx, CNRS, Ecole Polytechnique, Palaiseau, France — When an initially dry pile of micrometrical grains comes into contact with a liquid, one can observe different behaviors, function of the wetting properties. If the contact angle with the solid is low, the liquid will invade the pile (impregnation), while for higher contact angles, the grains will stay dry. We present an experimental study of this phenomenon: a dry pile of glass beads is deposed on the liquid surface, and we vary the contact angle of the liquid on the grains. We report a critical contact angle below which impregnation always occurs, and develop a model to explain its value. Different parameters modifying this critical contact angle are also investigated.

[1]Collaboration with Marco Ramaioli, Nestle Research Center, Lausanne, Switzerland

9:44AM M11.00009 Sweating Liquid Micro-Marbles: Drop wise condensation on hydrophobic particulate materials

MAHESH PANCHAGNULA, Indian Institute of Technology Madras, PRASAD BHOSALE, Department of Chemical Engineering, University of Washington, Seattle — Liquid marbles have been presented as important candidates in such important applications as gas sensing and bulk liquid transport since they were first proposed. In the current study, we present a remarkably simple self-assembly process driven by condensation on a nanoparticulate matrix giving rise to nearly monodisperse liquid marbles whose sizes can be controlled in the range of diameters from 2μm to 1000μm, monodisperse within a few micrometers in distribution width. We show that the primary mechanism causing the formation of liquid marbles is droplet nucleation followed by growth driven by condensation. Drop coalescence in dense droplet ensembles is the secondary mechanism, which attempts to destroy the distribution width controllability. It will also be demonstrated that coalescence dominated growth gives rise to a hitherto unreported growth law owing to the extremely high degree of mobility of embedded liquid marbles. Through a combination of understanding derived from these two physical processes, it will be shown that the proposed process provides control over both the mean marble diameter as well as the marble size distribution.

9:57AM M11.00010 DNS of thermocapillary flows based on two-scalar temperature representation

DIETER BOTHE, CHEN MA, Center of Smart Interfaces, Technische Universität Darmstadt, Germany — The direct numerical simulation (DNS) of thermocapillary two-phase flow with free deformable interface requires the solution of the two-phase Navier-Stokes equations in 3D together with the energy balance. We employ the sharp interface model which is solved using an extended volume of fluid method, where the discretization is based on Finite Volumes. The energy equation is given in temperature form, where the temperature field is represented by two scalars, one for each phase. This way the averaging over grid cells is confined to the individual phases and, hence, a smearing of the temperature gradient jump is avoided. Interpolation of the temperature within interfacial cells, exploiting the energy transmission condition, yields accurate temperatures at the interface, which is of utmost importance for the calculation of thermocapillary forces. Here the position and orientation of the interface is approximated by piecewise linear interface construction (PLIC). This method is applied to investigate liquid films on locally heated planar, respectively heated structured substrates. The approach allows for the numerical simulation of evaporating flows coupled with thermal Marangoni effects.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M12 Turbulence Simulation V 315

8:00AM M12.00001 Stochastic quantification of errors in large-eddy simulations of a spatially-evolving mixing layer, MARIA-VITTORIA SALVETTI, Aerospace Eng. Dept.- University of Pisa (Italy), MARCELLO MELDI, PIERRE SAGAUT, d' Alembert Inst. - UPMC Paris 6 (France) — The development of methodologies aimed at obtaining new insights in the behavior of the error in Large-Eddy Simulation (LES) has recently gained considerable attention. A possible approach to estimate the error at moderate computational cost is to combine the response surface methodology with the generalized Polynomial Chaos (gPC) theory, in which statistical information on the system response can be obtained by modeling the uncertain quantities through input random variables with given statistics. The gPC approach is used herein to quantify the error in LES of a spatially-evolving mixing layer and its sensitivity to different simulation parameters, viz. the grid stretching in the streamwise and lateral directions and the subgrid scale model constant. The error is evaluated with respect to the results of a highly-resolved LES and for different quantities of interest. The considered spatially-evolving flow is characterized by the progressive transition from a laminar regime, highly dependent on the inlet conditions, to a fully-developed turbulent one. Therefore the computational domain is divided in two different zones (inlet dependent and fully turbulent)and the gPC error analysis is carried out for these two zones separately. An optimization of the parameters is also carried out for both zones.

8:13AM M12.00002 Constrained Large Eddy Simulation of Separated Turbulent Flows, ZHENHUA XIA, YIPENG SHI, JIANCHUN WANG, ZUOLI XIAO, YANTAO YANG, SHIYI CHEN, State Key Laboratory for Turbulence and Complex System, College of Engineering, Peking University, China — Constrained Large-eddy Simulation (CLES) has been recently proposed to simulate turbulent flows with massive separation. Different from traditional large eddy simulation (LES) and hybrid RANS/LES approaches, the CLES simulates the whole flow domain by large eddy simulation while enforcing a RANS Reynolds stress constraint on the subgrid-scale (SGS) stress models in the near-wall region. Algebraic eddy-viscosity models and one-equation Spalart-Allmaras (S-A) model have been used to constrain the Reynolds stress. The CLES approach is validated *a posteriori* through simulation of flow past a circular cylinder and periodic hill flow at high Reynolds numbers. The simulation results are compared with those from RANS, DES, DDES and other available hybrid RANS/LES methods. It is shown that the capability of the CLES method in predicting separated flows is comparable to that of DES. Detailed discussions are also presented about the effects of the RANS models as constraint in the near-wall layers. Our results demonstrate that the CLES method is a promising alternative towards engineering applications.

8:26AM M12.00003 Dynamic Lagrangian model for LES on unstructured grids, KRISHNAN MAHESH, AMAN VERMA, University of Minnesota — We discuss a dynamic Lagrangian averaging approach applied in conjunction with the dynamic model for large-eddy simulation. Unlike Meneveau's Lagrangian dynamic model where the Lagrangian time scale contains an adjustable parameter θ, we propose a dynamic time scale based on a "surrogate-correlation" of the Germano-identity error (GIE). Also, a simple material derivative relation is used to calculate GIE at different events along a pathline instead of Lagrangian tracking. The absence of any multi-linear interpolation makes this approach particularly suitable for unstructured grids. The proposed model is applied to LES of turbulent channel flow at various Reynolds numbers and grid resolutions. Significant improvement over the dynamic Smagorinsky model is observed, especially at coarse resolutions. The model is also applied to external flow over a cylinder at high Reynolds numbers. This work was supported by the United States Office of Naval Research under ONR Grant N00014-08-1-0433.

8:39AM M12.00004 Dynamic time scale for the Lagrangian subgrid-scale model based on Rice's formula[1], CLAIRE VERHULST, CHARLES MENEVEAU, Mechanical Engineering and CEAFM, Johns Hopkins University — The dynamic formulation of Smagorinsky's subgrid-scale model for Large Eddy Simulations (LES) requires averaging to avoid instability due to extreme fluctuations. For complex-geometry flows a Lagrangian approach is often useful [see Meneveau, Lund, and Cabot, JFM 319 (1996)]. However, an ad-hoc choice of the relaxation timescale must be made, often based on resolved strain-rates and stresses at the grid- scale. Recently, Park and Mahesh [Phys. Fluids 21, 065106 (2009)] proposed the attractive notion of using statistics of the error signal itself to determine a timescale dynamically. We extend this approach by using Rice's formula to dynamically estimate the time between mean-crossings of the error signal and set the averaging timescale to be twice this value. The approach requires accumulating Lagrange-averaged square error and its time-derivative squared, which is done using the Eulerian approximation as proposed in the original model. For validation, LES of flow in a channel and through an array of cubes are compared with experimental results. Distributions of the dynamic coefficient, error, and dynamic timescale are shown as a function of distance from the wall. Computational efficiency and memory requirements are also discussed.

[1]Supported by NSF (GRFP and AGS-1045189).

8:52AM M12.00005 Convergence and scaling of large-eddy simulations of a turbulent free jet flow, HAIFENG WANG, STEPHEN POPE, Cornell University — A large set of large-eddy simulations (LES) is performed for a turbulent free jet flow with Reynolds number 21,000 to investigate systemically the convergence and scaling of the LES results with respect to the turbulence resolution scale (the filter width Δ) and the grid size h. Four convergence problems are considered: (a) convergence of the numerical error with h, for the Smagorinsky model with fixed Δ; (b) convergence of the Smagorinsky model error with Δ, for fixed h ($h \leq \Delta$); (c) convergence of the Smagorinsky model with $\Delta = h$; (d) convergence of the dynamic Smagorinsky model with $\Delta = h$. The convergence results are analyzed for the different LES quantities: the sub- filter eddy viscosity, the sub-filter shear stress, the resolved first, second and third order statistics. The scaling laws of the different LES quantities are analyzed based on the Kolmogorov energy-spectrum, and the LES convergence results agree with the scaling very well.

9:05AM M12.00006 Grid-point requirement for large eddy simulation: Chapman's estimation revisited[1], HAECHEON CHOI, Seoul National University, PARVIZ MOIN, Stanford University — Resolution requirements for large eddy simulation (LES), estimated by Chapman [AIAA J. Vol. 17, p. 1293 (1979)], are modified using accurate formulae for high Reynolds number boundary layer flow. This correction indicates that the number of grid points (N) required for wall-modeled LES is proportional to $Re_L^{2/7}$, but a wall-resolving LES requires $N \sim Re_L^{13/7}$, where L is the flat-plate length in the streamwise direction. The number of grid points required for the flow over an aircraft using LES with and without modeling the viscous wall region is estimated: the number of grid points for the wall-modeled LES is one to three orders of magnitude smaller than that for the wall-resolving LES, indicating the practical importance of wall modeling in LES for high Reynolds number flows.

[1]This work is conducted while HC takes a sabbatical year at Stanford University.

9:18AM M12.00007 Explicitly filtered LES on unstructured grids[1] , SANJEEB BOSE, PARVIZ MOIN, FRANK HAM, Center for Turbulence Research, Stanford University — Prior investigations using explicitly filtered LES have demonstrated that grid- independent solutions can be obtained by decoupling the filtering operator from the underlying grid. Dynamic, mixed SGS models were then derived from the explicitly filtered LES governing equations and showed good accuracy in coarse simulations of high Reynolds number wall-bounded flows. The explicitly filtered framework and SGS models are now implemented in a second order, unstructured, finite volume solver. Filtering on unstructured grids is decoupled from the mesh by utilizing differential filters. LES, using the proposed dynamic mixed models, of a Re=50000 rectangular duct flow (aspect ratio = 3.33) is performed. The grid is anisotropically refined in the near-wall region in the vicinity of the duct midplane away from the side walls. The regions of grid refinement are selected by processing the mean statistics of $\overline{u'_i}\overline{u'_i}$, which measures the smoothness of the LES solution with respect to the filter width. The overall resolution of the LES remains coarse ($\Delta x_f^+ \approx 150$, maximum $\Delta z_f^+ \approx 60$). Streamwise mean velocity profiles are predicted within a few percent of the experimental measurements of Kolade and Eaton (2010). Preliminary simulations of a three dimensional stalled diffuser will also be presented.

[1]Supported by the DOE CSGF (grant number DE-FG02-97ER25308)

9:31AM M12.00008 ABSTRACT WITHDRAWN —

9:44AM M12.00009 The effects of extended stencil sizes and mildly anisotropic grids on finite volume OLES , SIGFRIED HAERING, NICHOLAS MALAYA, JEREMY HIRA, ROBERT MOSER, University of Texas at Austin — Optimal large eddy simulations (OLES) utilize stochastic estimation to approximate the convective terms and produce statistically accurate LES. This formulation requires multi-point velocity correlations as input for the sub-grid model. Rather than relying on DNS data, these correlations can be completely determined from Kolmogorov inertial-range theory, small-scale isotropy, and the quasi-normal approximation. Initially, these models were developed for a particular stencil ($4 \times 1 \times 1$) on an isotropic grid, and produced accurate LES results. However, for OLES to become generally useful, it must be developed for anisotropic and inhomogeneous grids. Additionally, the optimal extent of correlation information to include must be determined. Here, we examine the effects of extending stencil sizes and introducing mild anisotropy. We present results from simulations characterizing the dependence of the model operators on a wide range of stencils and grid sizes. Modestly sized stencils are found to produce results nearly identical to the larger sets. These results provide evidence of ideal OLES operators for generic grids.

9:57AM M12.00010 Wavelet-based adaptive LES with explicit-filtering. , GIULIANO DE STEFANO, University of Naples, OLEG V. VASILYEV, University of Colorado at Boulder — Wavelet-based adaptive large-eddy simulation is a novel approach to the numerical simulation of turbulence, where the coherent energetic eddies are solved for, while modelling the influence of the less energetic coherent/incoherent background flow. The formal separation between resolved and unresolved turbulent velocity field is obtained by wavelet threshold filtering that is inherent to the adaptive wavelet collocation numerical method. A new explicit wavelet filtering strategy is introduced and tested, by considering two different filtering levels: the physical level, which controls the turbulence model, and the numerical level that is responsible for the accuracy of numerical calculations. The theoretical basis for wavelet-based adaptive large-eddy simulation with explicit filtering and consistent dynamic modelling is given. Some numerical experiments are presented for unsteady homogeneous turbulence, demonstrating the existence of grid-independent solutions.

Tuesday, November 22, 2011 8:00AM - 10:10AM —
Session M13 Multiphase Flow VI: Numerical III 316

8:00AM M13.00001 Dynamics of elastic particles in an elongational flow[1] , TONG GAO, HOWARD HU, PEDRO PONTE CASTANEDA, University of Pennsylvania — We consider dynamics of elastic particles in a steady elongational viscous flow under Stokes flow conditions. The particle is assumed to be an incompressible neo-Hookean elastic solid. A polarization technique based on Eshelby's problem in elasticity is used to describe the finite-strain, time-dependent response of the particle. Under simple flow conditions (e.g., with constant velocity gradient), a set of coupled, nonlinear, first-order ODEs is obtained for the evolution of the uniform stress fields in the particle, as well as for the shape and orientation of the particle. In the elongational flow, the particle deforms and rotates to align with the flow directions before reaching steady state. However, the steady-state solutions only exist in certain regimes where the deformation is not very large. In 2D, the closed-form analytical solutions can be solved directly; while in 3D, the solutions have to be solved numerically. The results of initially spherical shaped particles compare well with those of the classical work by Roscoe (1967).

[1]We acknowledge the Nano/Bio Interface Center at the University of Pennsylvania.

8:13AM M13.00002 Dynamics of elastic particles in a shear flow[1] , HOWARD HU, TONG GAO, PEDRO PONTE CASTANEDA, University of Pennsylvania — We study the dynamics of elastic particles in a simple shear flow under Stokes flow conditions. Both 2D and 3D situations are considered. The particle is assumed to be an incompressible neo-Hookean elastic solid. A polarization technique based on Eshelby's problem in elasticity is used to describe the finite-strain, time-dependent response of the particle. Under simple flow conditions (e.g., with constant velocity gradient), a set of coupled, nonlinear, first-order ODEs is obtained for the evolution of the uniform stress fields in the particle, as well as for the shape and orientation of the particle. Three types of particle motion which are typically seen in the vesicle motion under shear—steady-state, trembling and tumbling—are captured by changing the shear rate, the elastic modulus and the initial shape. The conditions that determine the trembling- to-tumbling transition, as well as the critical behavior of the particle near transition are explored. Additionally, the effective viscosity of a suspension of such soft particles under shear is also computed.

[1]We acknowledge the Nano/Bio Interface Center at the University of Pennsylvania.

8:26AM M13.00003 Rotation of triaxial particles in shear and strain: From Jeffery orbits and alignment to oscillations and chaos , FREDRIK LUNDELL, Royal Institute of Technology — The rotation of non-spherical particles is a cornerstone for the understanding of particle motion in flows laden with (non-spherical) particles. Here, the rotary motion of triaxial ellipsoids in linear shear and strain is studied theoretically and numerically under the assumption of Stokes flow ($Re = 0$). The particle motion is modeled by coupling the torques (Jeffery 1922) with the equations of rotation. The problem is governed by the relationships between the two shorter and the major axis together with the Stokes number, obtained as the Reynolds number times the particle/fluid density ratio. The general behaviour is that the particle ultimately rotates with the shortest axis aligned with the vorticity axis in shear flow. In linear strain, the particle aligns with the strain. However, the route and time to reach this state is distinctively different for different parameter combinations. Furthermore, the ultimate state is unstable for certain parameter combinations and the particle instead moves in a chaotic manner. The results, valid at $Re = 0$, shows a wide range of behaviors and provide a firm base for the understanding of particle motion also at higher Re.

8:39AM M13.00004 Numerical simulation on subcooled pool boiling bubble behavior , YASUO OSE, TOMOAKI KUNUGI, Kyoto University — In this study, it is focused on the clarification of the heat transfer characteristics of the subcooled pool boiling, the discussion on its mechanism, and the establishment of a boiling and condensation model for the direct numerical simulation on the subcooled pool boiling phenomena. In this paper, the boiling and condensation model has been improved by introducing the following models based on the quasi-thermal equilibrium hypothesis: (1) an improved phase-change model which consisted of the enthalpy method for the water-vapor systems, (2) a relaxation time derived by considering the unsteady heat conduction at the vapor-water interface. Then, unsteady three dimensional numerical simulations based on the MARS (Multi-interface Advection and Reconstruction Solver) with the improved boiling and condensation model were performed for the bubble departing process from the heated surface. The results of the numerical simulations were compared with the experimental data obtained by the high-speed camera (Phantom 7.1) with the Cassegrain optical system. As the results, the numerical results of bubble departure behavior from the heated surface showed in good agreement with the experimental observations quantitatively.

8:52AM M13.00005 Lattice Boltzmann Simulation of a Vapor Bubble in a 2D Microchannel , MICHAEL IKEDA, LAURA SCHAEFER, University of Pittsburgh — As energy densities in electronic devices rapidly increase, improved two-phase microchannel heat exchanger designs are of great interest. However, a better understanding of flow boiling in these regimes is needed. Experimental studies have thus far shown a great deal of variety in the flow patterns and instabilities that develop at the microscale level. Thus, numerical techniques capable of simulating such conditions are desirable. To this end, the behavior of a vapor bubble in a 2D microchannel is numerically analyzed. The kinetic lattice Boltzmann method is used over traditional CFD approaches due to its ability to capture the interfacial dynamics of multi-phase flow without the need for interface tracking algorithms. The single- component, multi-phase Shan-Chen model is utilized in conjunction with the passive scalar thermal approach, whereby the temperature field is passively advected by the hydrodynamics of the system. Advanced equations of state are implemented for the reduction of spurious currents and the recovery of realistic temperatures. The bubble dynamics are analyzed with varying Reynolds numbers and thermal boundary conditions.

9:05AM M13.00006 Numerical study on deformation and coalescence of droplets in a two-dimensional channel flow[1] , SUNGROK JUNG, MYOUNGHWAN CHO, Seoul National University, HYOUNGGWON CHOI, Seoul National University of Science and Technology, JUNGYUL YOO, Seoul National University — An improved level-set (LS) method is implemented to simulate the two-phase incompressible flow in two-dimensional channel considering the effect of interfacial tension. A mixed element is adopted, so that the Navier-Stokes equations are solved by using the q2q1 integrated finite element method (FEM), and the LS function is solved by using the q1q1 element. Direct approach method using geometric information is implemented instead of the conventional hyperbolic-type partial differential equation for reinitializing the LS function. The present code is verified by comparing the droplet movement and merging process with existing studies. It is shown that the computational results for deformation and migration of the droplets are in good agreement with those of the previous studies. In addition, the droplet merging process in straight and diverging channels is studied by using the present method. Comparing with the experimental data, the results of the present study shows similar tendencies for respective cases.

[1]Supported partially by Priority Research Centers Program through NRF funded by MEST (No. 0420-20100115), and the corresponding author was supported by Mid-Career Researcher Program through NRF grant funded by MEST (No. 2009-0079936), Republic of Korea.

9:18AM M13.00007 ABSTRACT WITHDRAWN —

9:31AM M13.00008 Effects of Surfactant on the Motion of Large Bubbles in a Capillary Tube[1] , METIN MURADOGLU, UFUK OLGAC, Koc University — The effects of surfactant on the Bretherton problem are studied computationally using both insoluble and soluble surfactant models. Emphasis is placed on the liquid film thickness between the bubble and the tube wall. We solve the evolution equations of the interfacial and bulk surfactant concentrations coupled with the incompressible Navier-Stokes equations. A non-linear equation of state is employed based on Langmuir adsorption. The numerical method is first validated for the clean bubble case and the results are found to be in a good agreement with the semi-analytical Taylor's law. Then the method is used to investigate the effects of insoluble and soluble surfactants on the film thickness for a wide range of governing non-dimensional numbers. It is found that both the insoluble and soluble surfactants have a thickening effect on the liquid film, which compares qualitatively well with both the experimental results and analytical predictions. Further computations are performed to examine the effects of non-dimensional numbers in the insoluble and soluble surfactant cases and it is found that elasticity, Damkohler and Peclet numbers have significant influence on the film thickness. Finally the computations are performed to examine the validity of insoluble surfactant for a wide range of governing nondimensional parameters.

[1]TUBITAK-108M238 and TUBA-GEBIP

9:44AM M13.00009 Detailed Numerical Simulation of Liquid Jet in Cross Flow Atomization with High Density Ratios[1] , SINA GHODS, MARCUS HERRMANN, Arizona State University — Atomization of a liquid fluid jet by a high speed cross-flowing gas has many applications such as gas turbines and augmentors. The mechanisms by which the liquid jet initially breaks up, however, are not well understood. Detailed numerical simulation can offer a better understanding of the underlying physical mechanisms that lead to the initial breakup of the injected liquid jet. In this work, we present detailed numerical simulation results of turbulent liquid jets injected into turbulent gaseous cross flows at varying momentum flux ratios and crossflow Weber numbers. We employ a finite volume, balanced force fractional step flow solver to solve the Navier-Stokes equations coupled to a Refined Level Set Grid method to follow the phase interface. To ensure discrete consistency between the solution of the conservative momentum equation and the level set based continuity equation, we employ a novel Rescaled Conservative Momentum Method. We analyze the impact of the previously-mentioned characteristic numbers on jet penetration, atomization mechanism, liquid mass flux distribution, and resulting drop size distribution and compare our numerical results to those obtained experimentally by Brown & McDonell (2006).

[1]This work was supported by NSF grant number CBET-0853627

9:57AM M13.00010 A fast Lagrangian tracking method capturing finite-size effects in particulate flows , MINGQIU WU, Universidad Carlos III, Madrid, Spain, JULIEN FAVIER, ALFREDO PINELLI, CIEMAT, Madrid, Spain, MODELING AND NUMERICAL SIMULATION GROUP TEAM — We present a new method to capture the finite-size effects induced by particles transported by a fluid flow, with a low computational cost compared to available fully-resolved methods, thus allowing to tackle configurations with high volume fractions of particles. The basic idea consists in tracking a source/sink of momentum occurring within a compact support of the mesh, centered on the particle and taking the form of a mollified Dirac kernel, or blob. In the spirit of the immersed boundary method, the shape and the intensity of the kernel are found by imposing appropriate reproducing conditions on the blob (to model accurately a Dirac pulse) and spreading on the mesh cells a volume force determined by the desired boundary condition. The particles occupy a finite-size volume of fluid, therefore introducing a two-way coupled behavior, for the computational cost of only one Lagrangian point. To build the blobs, we will either spread a zero-velocity condition at the blob center, or spread a zero-velocity condition averaged on the fluid parcel enclosed within the support. Both methods are discussed and validated by comparing with free falling fully-resolved particles, in 2D and 3D.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M14 Experimental Techniques IV 317

8:00AM M14.00001 High-Temperature Surface Thermometry Technique based on Upconversion Nano-Phosphors[1] , C. COMBS, N. CLEMENS, U. of Texas at Austin, X. GUO, H. SONG, H. ZHAO, K.K. LI, Y.K. ZOU, H. JIANG, Boston Applied Technologies Inc. —

Downconversion thermographic phosphors have been extensively used for high-temperature surface thermometry applications (e.g., aerothermodynamics, turbine blades) where temperature-sensitive paint is not viable. In *downconversion* techniques the phosphorescence is at longer wavelengths than the excitation source. We are developing a new *upconversion* thermographic phosphor technique that employs rare-earth-doped ceramics whose phosphorescence exhibit a strong temperature dependence. In the upconversion technique the phosphor is excited with near-IR light and emission is at visible wavelengths; thus, it does not require expensive UV windows and does not suffer from interference from background fluorescence. In this work the upconversion phosphors have been characterized in terms of their intensity, lifetimes and spectral content over a temperature range of 300K to 1500K. The technique has been evaluated for applications of 2D surface temperature measurements by using the total integrated intensity and the ratio of emission in different visible color bands. The results indicate that upconversion phosphor thermometry is a promising technique for making non-contact high-surface temperature measurements with good accuracy.

[1]Work supported by NASA under contract NNX11CG89P.

8:13AM M14.00002 Multi-Wavelength Optical Pyrometry Investigation for Turbine Engine Applications. , JORDI ESTEVADEORDAL, NIRM NIRMALAN, GUANGHUA WANG, GE Global Research, THERMAL SYSTEMS TEAM —

An investigation of optical Pyrometry using multiple wavelengths and its application to turbine engine is presented. Current turbine engine Pyrometers are typically broadband Si-detector line-of-sight (LOS) systems. They identify hot spots and spall areas in blades and bucket passages by detection of bursts of higher voltage signals. However, the single color signal can be misleading for estimating temperature and emissivity variations in these bursts. Results of the radiant temperature, multi-color temperature and apparent emissivity are presented for turbine engine applications. For example, the results indicate that spall regions can be characterized using multi-wavelength techniques by showing that the temperature typically drops and the emissivity increases and that differentiates from the emissivity of the normal regions. Burst signals are analyzed with multicolor algorithms and changes in the LOS hot-gas-path properties and in the suction side, trailing edge, pressure side, fillet and platform surfaces characterized.

8:26AM M14.00003 Development of Dual Luminescence Imaging for Capturing the Temperature Distribution of Supercooled-Water Droplets , AKIHITO AOKI, SHIGEO KIMURA, Kanagawa Institute of Technology, KATSUAKI MORITA, University of Tokyo, HIROTAKA SAKAUE, Japan Aerospace Exploration Agency —

Dual luminescent imaging technique is developed to capture the temperature field of supercooled-water droplets. This technique can be applied to obtain the temperature information for the icing in flights, power cables, architectures, and etc. It consists of two luminophores in water solution; one is sensitive to the temperature and the other is insensitive to the temperature. The former is used as a signal luminescence, and the other as a reference luminescence. Both are spectrally separated, which are captured by a hi-speed color camera. The temperature calibration in water below freezing point is shown. As a demonstration, the temperature field of a supercooled droplet on a plate with a different temperature is shown.

8:39AM M14.00004 Development and Testing of Temperature Sensitive Particles[1] , TRENTON WEST, University of Washington, CUN ZHU, YOUNAN XIA, Washington University in St. Louis, GAMAL KHALIL, DANA DABIRI, University of Washington —

Temperature sensitive particles can be used to seed a fluid flow, allowing for non-intrusive velocity and temperature field measurements. Using an avalanche photodiode (APD) with fiber optic setup, the emission intensity of luminescing samples is measured. Changes in the luminescence when subjected to a step temperature input within a shock tube lead to the determination of the time response. Furthermore, temperature and pressure sensitivity analysis is performed using a photomultiplier tube (PMT) survey apparatus. Results for various temperature sensitive particles are presented along with suggestions for application to experimental flows.

[1]This work is supported by the National Science Foundation (NSF/CBET-IDR- 0929864).

8:52AM M14.00005 Uncertainty analysis of particle image thermometry using individual thermo-liquid crystal tracers , RODRIGO SEGURA, CHRISTIAN CIERPKA, MASSIMILIANO ROSSI, CHRISTIAN KÄHLER —

An analysis of the benefits and challenges in calibrating temperature measurements by tracking the color fluctuations of individual thermo-liquid crystals is presented. Flow thermometry using liquid crystals has long been investigated for the evaluation of temperature fields but a reliable experimental uncertainty analysis is yet to be made for individual particles. A study was performed on the temperature response of individual tracer particles. The fluctuations in color response of adjacent crystals in a constant temperature environment were evaluated as well as the aging effects due to continuous illumination. The hysteresis of the color response to temperature gradients was also evaluated, as well as overheating and overcooling effects on individual particles. Flow thermometry has been widely used in the past to evaluate average temperature fields but a more precise characterization is possible by tracking the temperature of the individual tracers.

9:05AM M14.00006 Instantaneous Temperature Measurements using Constant-Voltage Anemometry , GENEVIEVE COMTE-BELLOT, Ecole Centrale Lyon, ARGANTHAEL BERSON, Durham University, PHILIPPE BLANC-BENON, EMMANUEL JONDEAU, LMFA, UMR CNRS 5509, Ecole Centrale Lyon —

Up to now, cold wires have been operated by constant-current anemometers, with a classic thermal inertia correction based on the mean value of the cold wire time constant [Lemay, Benaïssa & Antonia, Exp. Thermal & Fluid Sci, 2003, 27,133-143] or with a new correction method making use of the instantaneous value of the cold wire thermal lag [Berson, Poignand, Blanc-Benon & Comte-Bellot, Rev. Sci. Instrum. 2010, 81, 015102]. The latter correction method is applied to the constant-voltage anemometer and temperature measurements are presented for the first time with such a device. Two constant-voltage anemometers are used for the instantaneous measurement of temperature fluctuations in unsteady flows. The first one is a new prototype elaborated by Tao Systems Inc. and adapted to fine wires with a resistance between 30 and 100 ohms. It operates a cold wire whose resistance varies with the temperature of the surrounding fluid. The second anemometer is a commercial system by the same company. It operates a hot wire, from which the instantaneous effect of the thermal inertia of the cold wire is determined. Results are presented for two flows: (i) a heated turbulent jet and (ii) an acoustic standing wave inside a resonator where flow reversal occurs.

9:18AM M14.00007 Toluene laser-induced fluorescence imaging of compressible flows in an expansion tube[1]

V.A. MILLER, M. GAMBA, M.G. MUNGAL, R.K. HANSON, Stanford University, K. MOHRI, C. SCHULZ, University of Duisburg-Essen — Laser-induced fluorescence (LIF) imaging using toluene as a tracer molecule has been developed for high-speed, low-to-moderate enthalpy conditions in the Stanford 6-inch Expansion Tube. The approach is demonstrated on three canonical compressible flow configurations: (i) supersonic flow over a 20° wedge, (ii) around a cylinder, and (iii) a supersonic boundary layer. Under constant-pressure conditions, toluene LIF offers unique sensitivity to temperature and can therefore be used as an accurate thermometry diagnostic for supersonic flows; on the other hand, for variable-pressure flow fields (e.g., flow around a blunt body), toluene LIF imaging is demonstrated to be an effective flow visualization tool. The three configurations selected demonstrate the diagnostic in these two capacities. For all configurations considered in the study, toluene (0.6% by volume) is seeded into a nitrogen freestream at a Mach number $\sim$ 2.2, T $\sim$ 500K, and p $\sim$ 1.5 bar. A frequency-quadrupled pulsed Nd:YAG laser is used to excite the tracer, and the resulting fluorescence is captured by an ICCD camera. Synthetic fluorescence signals from CFD solutions of each case have been computed and compare favorably to measured signals.

[1]Sponsored by DoE PSAAP at Stanford University.

9:31AM M14.00008 PIVT Investigation on Film Cooling in Turbine Blade

REBECCA MALISH, JORDI ESTEVADEORDAL, RON BUNKER, GE GLOBAL RESEARCH THERMAL SYSTEMS TEAM — An investigation using particle image velocimetry and thermography (PIVT) on the film cooling location of turbine blades in a cascade is presented. The laser scattering from seeding particles is captured with double exposure PIV camera, and fluorescence emissions are captured with ICCD camera for ratio thermometry. The technique allows capturing the velocity and temperature distributions of the particles at varying blowing ratios. The seeding particles were immersed in two locations of the cascade, the first was upstream of the blade; the second was inside the blade to allow for mixing with the coolant. The combined velocity and temperature maps allow understanding the film holes cooling effectiveness and clearly identify areas and holes that are more effective at cooling. PIVT gives a two-dimensional measurement than other techniques cannot provide and it is able to map the velocity and temperature in various viewing flow areas with the seeding particles.

9:44AM M14.00009 Glowing Droplets in Turbulence[1]

HUMBERTO BOCANEGRA EVANS, Eindhoven University of Technology, THANJA LAMBERTS, Radboud University Nijmegen, NICO DAM, WILLEM VAN DE WATER, Eindhoven University of Technology, JM BURGERSCENTRUM COLLABORATION — Preferential concentration is a process in which turbulence unmixes a dispersion of droplets. It is most prominent at the smallest length and time scales. We select droplets by phosphorescence tagging in a zero mean flow turbulence experiment. The droplets are made out of a Europium-based solution, which has a phosphorescent lifetime of approximately 1 ms. A cylinder-shaped cloud is tagged in a weakly focused laser beam. The evolution of the cloud in the turbulent flow is recorded using an intensified high-speed camera. Through this method both, the advection of the tagged region and the evolution of the droplet concentration, can be monitored. The phosphorescent cloud is subjected to a turbulent flow by means of eight synthetic jets placed on the truncated corners of a cubic chamber. The turbulent flow is highly isotropic and has Re_λ=444, with Kolmogorov length– and timescale η=79 μm and τ_k=406 μs, respectively. We will present a proof of principle of this novel technique.

[1]This project is funded by Stichting voor Fundamenteel Onderzoek der Materie (FOM)

9:57AM M14.00010 Selection and Testing of Pressure and Temperature Sensitive Dyes for 2-D Flow Characterization via Synthesized Microbeads[1]

ALEX PEREZ, University of Washington, CUN ZHU, YOUNAN XIA, Washington University in St. Louis, GAMAL KHALIL, DANA DABIRI, University of Washington — Airborne temperature and pressure sensitive microbeads provide a vehicle with which to conduct two-dimensional flow characterization. An array of temperature and pressure sensitive dyes have been synthesized with microbeads (of silica, polystyrene, and polydimethylsiloxane) for this purpose. These microbeads were evaluated based on emission spectra, pressure response (0-760 torr), temperature response (5-45°C), and response time. Work will be presented showing the various combinations of dyes and microbead materials, as well as the testing process and examples of future application.

[1]This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. #DEG-0718124, as well as National Science Foundation Grant No. NSF/CBET-IDR- 0929864.

Tuesday, November 22, 2011 8:00AM - 10:10AM – Session M16 Free-Surface Flows III 319

8:00AM M16.00001 Numerical Simulation of the "Fluid Mechanical Sewing Machine"

PIERRE-THOMAS BRUN, Institut Jean le Rond dAlembert/Lab. FAST, UPMC, Univ Paris-Sud, CNRS, BASILE AUDOLY, Institut Jean le Rond dAlembert, UPMC, CNRS, NEIL RIBE, Lab FAST, UPMC, Univ Paris-Sud, CNRS — A thin thread of viscous fluid falling onto a moving conveyor belt generates a wealth of complex "stitch" patterns depending on the belt speed and the fall height. To understand the rich nonlinear dynamics of this system, we have developed a new numerical code for simulating unsteady viscous threads, based on a discrete description of the geometry and a variational formulation for the viscous stresses. The code successfully reproduces all major features of the experimental state diagram of Morris et al. (Phys. Rev. E 2008). Fourier analysis of the motion of the thread's contact point with the belt suggests a new classification of the observed patterns, and reveals that the system behaves as a nonlinear oscillator coupling the pendulum modes of the thread.

8:13AM M16.00002 Frequency structure of the nonlinear instability of a dragged viscous thread

STEPHEN W. MORRIS, Dept. of Physics, University of Toronto, ROBERT L. WELCH, Dept. of Physics, McGill University, BILLY SZETO, Dept. of Physics, University of Toronto — A thread of viscous fluid falling onto a moving belt exhibits a spectacular variety of modes of motion as the belt speed and nozzle height are varied [1]. For modest nozzle heights, four clear regimes are observed. For large belt speed, the thread is dragged into a stretched centenary configuration which is confined to a plane. As the belt speed is lowered, this exhibits a supercritical Hopf bifurcation to a meandering mode [2]. At very low belt speeds, the motion resembles the usual coiling motion of a viscous thread falling on a stationary surface. In between the meandering and coiling regimes, a window of novel multifrequency motion, previously called "figures of eight" is found. We examined the longitudinal and transverse motion of the thread in all these states, using an automated apparatus that allows a detailed exploration of the parameter space. We found that the multifrequency window is characterized by a complex pattern of motion whose main frequencies are locked in a 3:2 ratio. This motion appears and disappears with finite amplitude at sharp bifurcations, without measurable hysteresis.

[1] S. Chiu-Webster and J. R. Lister, J. Fluid Mech., 569, 89 (2006).
[2] S. W. Morris, J. H. P. Dawes, N. M. Ribe and J. R. Lister, Phys. Rev. E, 77, 066218 (2008).

8:26AM M16.00003 Liquid supercoiling , NEIL RIBE, Laboratoire FAST, Orsay, France, MEHDI HABIBI, HOSSEIN HOSSEINI, MOHAMMAD HASSAN KHATAMI, Institute for Advanced Studies in Basic Sciences, Zanjan, Iran — Supercoiling is defined as the large-scale secondary coiling of a slender body that is already coiled at a smaller scale (e.g., telephone cords and DNA strands). We demonstrate experimentally a novel fluid-mechanical form of supercoiling that occurs in the context of the familiar "liquid rope coiling" instability of a thin thread of viscous fluid falling onto a rigid surface. Under appropriate conditions, the coiling instability generates a tall pile of coils in the form of a hollow cylindrical column, which in turn becomes unstable to a secondary coiling instability with a frequency $\approx 10\%$ of the primary one. To place this phenomenon in a broader context, we determine experimentally the phase diagram for the different possible behaviors of the thread (stagnation flow, simple coiling, rotatory folding, periodic column collapse, supercoiling) in the space of the fluid viscosity, the flow rate, and the fall height. We formulate a mathematical model for supercoiling by combining a thin-shell description of the column wall with a slender-thread description of the column as a whole. This leads to a set of coupled ordinary differential equations in one space dimension (the arclength along the axis of the coiling column) that we solve numerically using a continuation method. A comparison of the predicted and observed frequencies of secondary coiling will be shown.

8:39AM M16.00004 Shape of a viscous curtain , MARC RABAUD, Univ. Paris-Sud 11 — When a viscous liquid springs out of a horizontal circular tube, at large enough flow rate the jet progressively takes the shape of a thin vertical curtain limited by two approximately parabolic rims. We will describe the shapes observed depending on the experimental conditions.

8:52AM M16.00005 On the the Contact Lens Problem: Modeling Rigid and Elastic Beams on Thin Films , PHILIPPE TRINH, Princeton University, STEPHEN WILSON, Strathclyde University, HOWARD STONE, Princeton University — Generally, contact lenses are prescribed by the practitioner to fit each individual patient's eye, but these fitting-philosophies are based on empirical studies and a certain degree of trial-and-error. A badly fitted lens can cause a range of afflictions, which varies from mild dry-eye-discomfort, to more serious corneal diseases. Thus, at this heart of this problem, is the question of *how a rigid or elastic plate interacts with the free-surface of a thin viscous film*. In this talk, we present several mathematical models for the study of these plate-and-fluid problems. Asymptotic and numerical results are described, and we explain the role of elasticity, surface tension, viscosity, and pressure in determining the equilibrium solutions. Finally, we discuss the implications of our work on the contact lens problem, as well as on other coating processes which involve elastic substrates.

9:05AM M16.00006 Numerical evidence for formation of "resonant" coherent structures by an ensemble of small rigid particles in thermocapillary flows , DENIS MELNIKOV, DMITRI PUSHKIN, VALENTINA SHEVTSOVA, Microgravity Research Center, Free University of Brussels (ULB) — The effect of formation of coherent particulate accumulation structures (PAS) in a thermocapillary flow was discovered more than a decade ago. It happens in regimes of the flow that are characterized by a hydrothermal wave travelling in the azimuthal direction. Those structures are dynamic and rotate azimuthally together with the travelling wave. They are appearing as a result of ubiquitous in nature nonlinear phenomenon of phase locking, when the turnover particle motion is synchronized with the rotating wave. Synchronization of the particles with the wave most commonly ends up in resonance leading to the PAS mode coinciding with that of the wave. Other resonance modes of PAS, however, may occur at the same time. We present numerical evidence for such structures.

9:18AM M16.00007 Holy balls! , TADD TRUSCOTT, Brigham Young University, JESSE BELDEN, Naval Undersea Warfare Center — Why can some balls walk on water while others cannot? We investigate the rebound dynamics of elastic spheres impacting on a free-surface. Several variables determine whether or not a sphere will bounce when impacting a free-surface including velocity, impact angle, size and elasticity. Stiff elastic spheres, such as a racquetball, successfully skip at low impact angles and high velocities, but tend not to bounce when the impact angle becomes too large. However, the more compliant Waboba® (WAter BOuncing BAll) bounces marvelously even at very high impact angles. Unlike a stiffer ball, the Waboba® flattens out quickly as it is forming a cavity. The cavity lip forms a ramp and the flattened ball then skips off the water surface. We demonstrate how this phenomenon surprisingly resembles a skipping stone. Using high-speed video we explore the rebound dynamics for various values of elasticity, velocity, angle and size and determine when an object will bounce off the water surface.

9:31AM M16.00008 The 2D Selective Withdrawal Transition: Analogies with MEM Systems , STUART KENT, SHANKAR VENKATARAMANI, University of Arizona — Free boundaries in selective withdrawal systems have been observed undergoing topological transitions through apparently-singular steady states as the withdrawal rate is increased. We transfer the study of this transition to a simpler class of differential equations that are analogous to those found in the study of microelectromechanical (MEM) systems. In true electrostatic free boundary problems, the electrostatic pressure varies as the square of the electric field. Artificially replacing this quadratic dependence with a linear dependence that matches the linear relationship between a Newtonian fluid stress field and velocity gradient components, we obtain a good model for the fluid system with fewer degrees of freedom. By first considering a restricted family of one dimensional boundaries related to two-parameter conformal maps, we aim to identify the mechanisms that control the boundary breakdown in two dimensions.

9:44AM M16.00009 Convection, evaporation, and condensation of binary fluids in confined geometries[1] , ROMAN GRIGORIEV, TONGRAN QIN, YAOFA LI, BENJAMIN CHAN, MINAMI YODA, Georgia Tech — Phase change has a major effect on convection in liquid layers with a free surface. Significant latent heat generated at the free surface as a result of phase change can dramatically alter the interfacial temperature, inducing thermocapillary stresses. For binary fluids, differential evaporation leads to a variation in the concentration, and hence, induces solutocapillary stresses. This talk describes numerical and experimental studies of convection in alcohol-water mixtures due to a horizontal temperature gradient in the presence of phase change. Evaporation and condensation is known to be a notoriously difficult problem to model due to a poorly defined vapor transport problem which is strongly influenced by the presence/absence and flows of non-condensable gases (e.g., air). This issue is addressed by using a sealed cuvette heated at one end and cooled at the other. Both numerics and experiments show that, by adding or removing air from the cuvette, the direction of flow in a liquid layer covering the bottom of the cell can be reversed by emphasizing either thermocapillary or solutocapillary stresses.

[1]Supported by ONR

9:57AM M16.00010 A new mechanism for atomization and the primary instability in liquid-gas mixing layers , STEPHANE ZALESKI, DALEMBERT, Universite Pierre et Marie Curie, Paris 6, ALAIN CARTELLIER, LEGI, CNRS and UJF, Grenoble, DANIEL FUSTER, JÉRÔME HOEPFFNER, DALEMBERT, Universite Pierre et Marie Curie, Paris 6, JEAN-PHILIPPE MATAS, LEGI, CNRS and UJF, Grenoble — We investigate numerically and experimentally the appearance of instabilities in two planar coflowing liquids sheets. As a function of the momentum ratio $M = \rho_G U_G^2 / \rho_L U_L^2$, two different regimes are distinguished. At low momentum ratios the frequency of the waves appearing in the primary atomization region is influenced by the liquid velocity, whereas another asymptotic regime is obtained for large momentum ratios. In this regime, the gas velocity and the ratio between the gas boundary layer and the thickness of the separator plate influence the observed frequency. The low M, liquid-dominated regime appears to be a noise amplifier, while the gas-dominated, large M regime displays the characteristics of a global mode. Current computational results are in agreement with experimental observations. These results are compared to the predictions of linearized stability theory. We discuss both the inviscid linearized stability theory and the viscous, Orr-Sommerfeld stability theory, and conjecture that the viscous stability theory is valid in the low M regime.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M17 Reacting Flows IV: LES of Turbulent Reacting Flows 320

8:00AM M17.00001 Turbulence-Radiation Interactions in Large-Eddy Simulations of Non-premixed Turbulent Jet Flames , D.C. HAWORTH, The Pennsylvania State University, A. GUPTA, Rolls-Royce, M.F. MODEST, University of California, Merced — Large-eddy simulation (LES) has been performed for four piloted nonpremixed turbulent jet flames: a nonluminous laboratory-scale flame, a luminous (sooting) laboratory-scale flame, a nonluminous larger-scale flame, and a luminous larger-scale flame. These flames range from optically thin to moderately optically thick. The simulations feature a transported composition filtered density function (FDF) method to account for the influence of subfilter-scale fluctuations in composition and temperature on resolved chemical reaction rates and radiative emission, and a photon Monte Carlo (PMC) method with line-by-line spectral resolution to model radiation heat transfer including re-absorption. New optimized FDF/PMC algorithms and parallelization strategies have been developed, such that fully coupled LES/FDF/PMC is computationally tractable. The model fully captures both emission and absorption turbulence-radiation interactions (TRI), and is exercised to quantify the contributions of subfilter-scale TRI. For mesh resolutions that are representative of those used in typical turbulent combustion LES studies (approximately 85 percent of the turbulence kinetic energy is resolved), subfilter-scale emission TRI are found to be important in all cases and subfilter-scale absorption TRI are negligible.

8:13AM M17.00002 A fractal dynamic SGS combustion model for turbulent premixed flames , ITARU YOSHIKAWA, YOUNGSAM SHIM, MAMORU TANAHASHI, TOSHIO MIYAUCHI, Tokyo Institute of Technology — A dynamic subgrid scale (SGS) combustion model for large eddy simulation (LES) of turbulent premixed combustion has been developed based on the fractal characteristics of turbulent premixed flames and the scale separation of turbulence. This model locally predicts the fractal dimension of a flame surface and the local flame surface area. To give the lower limit of the flame surface area, the contribution of dilatation across the flame is also considered. The local turbulent flame speed is then determined supposing the flamelet concept. In the model formulation, the Kolmogorov length scale is described with the grid scale (GS) variables to yield the inner-cutoff of the flame surface. It is achieved by assuming that, in high Reynolds number turbulence, most of the turbulence energy is dissipated in the SGS level and the dissipation balances with the energy production that is given by an SGS turbulence model. A series of static tests have been performed to examine the model by filtered data of direct numerical simulation (DNS). Gaussian or tophat filters with different widths within the range of the inner- and outer-cutoffs of the flame surface are applied as the LES and the test- filters. The results show that the predicted flame surface area agrees well with that extracted from the DNS data regardless of the filter type and the widths.

8:26AM M17.00003 Large-eddy simulation of a three-stream MILD combustion system , JIAN ZHANG, Institute of Mechanics, CAS, MATTHIAS IHME, Department of Aerospace, University of Michigan, Ann Arbor, GUOWEI HE, Institute of Mechanics, CAS — Large-eddy simulations (LES) of a three-stream burner system are performed. This burner is operated in the so-called moderate and intense low-oxygen dilution (MILD) combustion regime. An extended flamelet/progress variable (FPV) model is utilized, in which an additional scalar is introduced in order to account for the mixing between the three reactant streams. LES-calculations of three different operating conditions are performed, corresponding to increased levels of oxygen-dilution in the vitiated coflow. The extended FPV model accurately predicts effects of the oxygen-dilution on the flame-structure and heat-release, and model-predictions for temperature and major and minor species are in good agreements with the measurements.

8:39AM M17.00004 Large eddy simulation of an experimental partially-premixed flame configuration using a dynamic nonequilibrium model for subfilter scalar dissipation rate , COLLEEN M. KAUL, The University of Texas at Austin; Center for Turbulence Research, Stanford University, VENKAT RAMAN, The University of Texas at Austin — Accurate prediction of nonpremixed turbulent combustion using large eddy simulation (LES) requires detailed modeling of the mixing between fuel and oxidizer occurring at scales not resolved by the LES filterwidth. The scalar dissipation rate, a critical quantity for describing the rate of small scale mixing in conserved scalar combustion models, is dominated by its subfilter component. Nonequilibrium models for the subfilter scalar dissipation rate, which do not assume a local balance between production and dissipation of subfilter scalar variance, typically contain a model coefficient whose optimal value is unknown a priori for a given simulation. Furthermore, conventional dynamic procedures are not useful for estimating its value. An alternative dynamic modeling approach for the model coefficient has been developed based on the transport equation for subfilter scalar variance and validated in a priori tests. Here, the new dynamic nonequilibrium subfilter scalar dissipation rate model is used for simulation of an experimental partially-premixed flame configuration. Results obtained with the new model are compared to predictions using an equilibrium modeling approach for dissipation.

8:52AM M17.00005 LES of Swirling Reacting Flows via the Unstructured scalar-FDF Solver , NASEEM ANSARI, University of Pittsburgh, ANSYS Inc., PATRICK PISCIUNERI, University of Pittsburgh, PETER STRAKEY, National Energy Technology Laboratory, PEYMAN GIVI, University of Pittsburgh — Swirling flames pose a significant challenge for computational modeling due to the presence of recirculation regions and vortex shedding. In this work, results are presented of LES of two swirl stabilized non-premixed flames (SM1 and SM2) [1] via the FDF methodology. These flames are part of the database for validation of turbulent-combustion models [1]. The scalar-FDF is simulated on a domain discretized by unstructured meshes, and is coupled with a finite volume flow solver. In the SM1 flame (with a low swirl number) chemistry is described by the flamelet model based on the full GRI 2.11 mechanism. The SM2 flame (with a high swirl number) is simulated via a 46-step 17-species mechanism. The simulated results are assessed via comparison with experimental data.

[1] Sandia National Laboratories, TNF Workshop Website, Bluff-Body Flames, www.ca.sandia.gov/TNF/bluffbod.html, 2011.

9:05AM M17.00006 Data-Based Optimum Strategies for Combustion Large-Eddy Simulation , HESSAM MIRGOLBABAEI, TAREK ECHEKKI, North Carolina State University — In turbulent combustion, moment-based methods have enjoyed widespread use for solving turbulent combustion flows. In these methods, the combustion process is predicted using transport equations for moments (e.g. the mixture fraction) with additional closure needed to determine other thermochemical scalars. The choice of moments is driven by both intuition and experience; but, there is a growing need for optimum strategies to determine these moments. In this work, re-parameterization of the thermochemical state space of combustion flows is performed based on one- dimensional turbulence (ODT) simulations. The re-parameterization identifies optimum transported moments and tabulate key terms in their transport equations. The approach is implemented using principal component analysis (PCA) as a first step towards the evaluation of an optimum set of moments (the principal components) that can represent the thermochemical scalars in the data. Parameters from the PCA analysis can be used to evaluate key terms in the transport equation for these moments. These terms are tabulated using artificial neural network (ANN).

9:18AM M17.00007 A wavelet-based scheme for coupling large-eddy simulation with the one-dimensional turbulence model , YUQIANG FU, TAREK ECHEKKI, North Carolina State University — A wavelet-based scheme for coupling large-eddy simulation (LES) with the one-dimensional turbulence (ODT) model is developed. The LES-ODT approach is a multiscale framework for the simulation of turbulent reacting flows. It is based on the implementation of two solutions: 1) LES for continuity and momentum and 2) ODT for momentum and reactive scalars. Capturing large-scale transport for momentum and scalars is an integral component of the LES-ODT framework The objectives of the wavelet-based scheme are 1) to compound the large-scale velocity field from LES and ODT and 2) to dynamically determine a model parameter in ODT, which governs subgrid scale (SGS) stresses and scalar fluxes. The model is based on the compound wavelet matrix (CWM), which substitutes large-scale physics from LES onto the ODT solution, while maintaining the ODT residual SGS contribution intact. The ODT parameter, which is responsible for controlling the rate of SGS momentum and scalar transport in ODT is dynamically evaluated by comparing wavelet spectra from the ODT and LES solution around the filter cut-off scale. Different strategies are investigated to efficiently evaluate this parameter from an initial guess. The wavelet transform has unique features that enable an efficient and robust implementation of both compounding and ODT parameter estimation.

9:31AM M17.00008 Large-eddy simulation/PDF modeling of a non-premixed CO/H_2 temporally evolving jet flame[1] , YUE YANG, Cornell University and Sandia National Laboratories, HAIFENG WANG, STEPHEN B. POPE, Cornell University, JACQUELINE H. CHEN, Sandia National Laboratories — We report a large-eddy simulation (LES)/probability density function (PDF) study of a non-premixed CO/H_2 temporally evolving planar jet flame at Re = 9079 and Da = 0.011 with skeletal chemistry. The flame exhibits strong turbulence-chemistry interactions resulting in local extinction followed by re-ignition. In this study, the filtered velocity field in LES is computed using the NGA code (Desjardins et al., 2008) and the PDF transported equations with the modified Curl's mixing model are solved by the new highly-scalable HPDF code (Wang and Pope, 2011) with second order accuracy in space and time. The performance of the hybrid LES/PDF methodology is assessed through detailed a posteriori comparisons with DNS of the same flame (Hawkes et al., 2007). The comparison shows good agreement of the temporal evolution of the temperature and mass fractions of major chemical species, as well as the prediction of local extinction and re-ignition. In addition, the effects of the subgrid scale model, the mixing model, and grid resolution on turbulence-chemistry interactions are investigated to improve the capabilities of LES/PDF.

[1]Supported in part by the CEFRC funded by the DOE

9:44AM M17.00009 LES/PDF Approach for Modeling Soot Formation in Turbulent Flames , PRATIK DONDE, VENKAT RAMAN, Department of Aerospace Engineering and Engineering Mechanics, The University of Texas at Austin, MICHAEL MUELLER, Department of Mechanical Engineering, Stanford University, HEINZ PITSCH, Institute for Combustion Technology, RWTH Aachen University — In large eddy simulation based modeling of soot evolution in turbulent flames, the correlations between the gas phase scalars and the soot particles need to be modeled. Typically, the soot population is described in terms of a number density function, which is further parameterized using a finite set of moments. Modeling soot-gas phase correlations essentially implies modeling of the one-time one-point probability density function (PDF) of gas phase scalars and the soot moments. Typically, presumed functional forms for this joint-PDF are used. In this work, a transport equation for the PDF is directly evolved using a Monte-Carlo approach. Simulation of experimental flame configurations including a piloted methane flame and a bluff-body stabilized flame are used to compare the full PDF approach with presumed PDF models.

9:57AM M17.00010 Large Eddy Simulation Modeling and Flamelet Analysis of a Jet in Cross Flow , WAI LEE CHAN, University of Michigan, HEMANTH KOLLA, Sandia National Laboratories, MATTHIAS IHME, University of Michigan, JACQUELINE CHEN, Sandia National Laboratories — Jet in cross flow (JICF) configurations are frequently used as fuel injection strategies in combustion systems, such as gas turbines, boilers, and high-speed propulsion systems. Recently, direct numerical simulations (DNS) have been performed to investigate flame-stabilization mechanisms in a reactive JICF. By utilizing this DNS-database, fundamental modeling assumptions of flamelet-based large eddy simulation (LES) combustion models for application to JICF are evaluated. To this end, a priori and a posteriori studies are performed using steady and unsteady flamelet models to isolate and model combustion flow processes that control transient ignition events and flame stabilization.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M18 Acoustics I: Scattering, Jets and Cavity Flows 321

8:00AM M18.00001 Large-Eddy Simulations of Supersonic Heated Jets , JUNHUI LIU, K. KAILASANATH, Naval Research Lab., NICK HEEB, EPHRAIM GUTMARK, University of Cincinnati — Large-eddy simulations of imperfectly expanded jet flows from a convergent-divergent nozzle with a heated (600K) and a cold jet conditions have been carried out. Mach wave radiation is present outside the jet core and the heated jet shows a much larger density shear-layer spreading than the cold jet. The potential core length is shortened, but the shock-cell shape and shock-cell spacing remain similar to those shown in the cold jet. It is found that the Crocco-Busemann equation captures well the correlation between the temperature and the axial velocity in the cold jet, but it underestimates the nonlinear variations in the shock-containing region in the heated jet. Temperature effect has more impact on the high-frequency components of pressure fluctuations near the nozzle exit, but the impact moves to low- and mid-frequency ranges further downstream. The convection speed of the near-field pressure waves increases with the temperature. The heated jet is found to be more sensitive to the inflow pressure perturbations than the cold jet.

8:13AM M18.00002 Parabolized stability equation (PSE) models for the prediction of mixing noise in turbulent jets: nonlinearity and comparison with experiments[1] , TIM COLONIUS, DANIEL RODRIGUEZ, California Institute of Technology, ARNAB SAMANTA, Indian Institute of Science, Bangalore, ANDRE CAVALIERI, PETER JORDAN, Institute Pprime, Poitiers, France, CALIFORNIA INSITUTE OF TECHNOLOGY TEAM, INDIAN INSTITUTE OF SCIENCE COLLABORATION, INSTITUTE PPRIME, POITIERS TEAM — The wavepackets responsible for the generation of the dominant low-frequency noise of turbulent round jets are modeled via PSE. Previous experience shows that wavepackets delivered by linear PSE are in agreement with pressure fluctuations on a near-field microphone array. Nonlinear interactions between PSE modes is the subject of the present effort. Pressure fluctuations in the near-field and hot wire measurements at the jet centerline are used to determine initial amplitudes for the PSE computations, and then perturbed randomly to introduce some degree of nondeterminism. A projection technique is used to compare the noise radiated to the far-field by wavepackets with experiments. First results indicate that nonlinearity is not a determinant factor in the wavepackets evolution, especially for unheated, low Mach number jets, while small variations in the initial conditions might introduce $O(1)$ changes.

[1]Support from NAVAIR through TTC Tech., Inc. and Cascade, Inc.

8:26AM M18.00003 Parabolized stability equation (PSE) models for the prediction of mixing noise in turbulent jets: Comparison with Large Eddy Simulation[1] , DANIEL RODRIGUEZ, TIM COLONIUS, California Institute of Technology, ARNAB SAMANTA, Indian Institute of Science, Bangalore, YASER KHALIGHI, CASCADE Technologies Inc, CALIFORNIA INSITUTE OF TECHNOLOGY TEAM, INDIAN INSTITUTE OF SCIENCE TEAM, CASCADE TECHNOLOGIES INC TEAM — The wavepacket structures responsible for the generation of the dominant low- frequency noise of turbulent round jets are modeled via PSE. To deliver meaningful results, PSE models require an accurate description of the disturbance conditions in the vicinity of the nozzle. The nature of the unforced turbulent flow avoids the existence of a single, deterministic initial condition. A high fidelity database obtained by Large Eddy Simulation is used to determine the initial conditions for the PSE modes: the total length of the simulation is split into several realizations, and Linear Stability Theory based biorthogonal decomposition is employed to initialize the PSE with the adequate wavepacket. The PSE solution corresponding to each realization is then compared with the wavepackets educed from the LES dataset. The ensemble of the realizations is considered then to study the effect of nondeterministic conditions on the wavepacket evolution.

[1]Support from NAVAIR through TTC Tech., Inc. and Cascade, Inc.

8:39AM M18.00004 Modeling unsteady sound refraction by coherent structures in a high-speed jet[1] , PINQING KAN, JACQUES LEWALLE, Syracuse University — We construct a visual model for the unsteady refraction of sound waves from point sources in a Ma = 0.6 jet. The mass and inviscid momentum equations give an equation governing acoustic fluctuations, including anisotropic propagation, attenuation and sources; differences with Lighthill's equation will be discussed. On this basis, the theory of characteristics gives canonical equations for the acoustic paths from any source into the far field. We model a steady mean flow in the near-jet region including the potential core and the mixing region downstream of its collapse, and model the convection of coherent structures as traveling wave perturbations of this mean flow. For a regular distribution of point sources in this region, we present a visual rendition of fluctuating distortion, lensing and deaf spots from the viewpoint of a far-field observer.

[1]Supported in part by AFOSR Grant FA-9550-10-1-0536 and by a Syracuse University Graduate Fellowship.

8:52AM M18.00005 Noise source characterization from far-field data in high-speed jets[1] , JACQUES LEWALLE, KERWIN R. LOW, MARK N. GLAUSER, Syracuse University — We analyze pressure data from 3 microphones in the far-field coherent noise cone of a Ma=0.6 jet. 3500 individual sources are identified in the time-frequency domain, and some of their properties are extracted from the wavelet coefficients: magnitude, frequency, time of arrival. The sources' signature therefore includes the lags between the sound arrival times at the microphones. Based on a modeled mean jet velocity field, we calculate the refracted acoustic paths and propagation times from a grid of source locations to the microphones. The excellent agreement between measured and calculated lags provides a mapping between the measured lags and the approximate source locations. For our catalog of sources, we report on some statistics of source properties as a function of their location in the near-jet's shear layer and developing region beyond the end of the potential core. The ability to narrow the search (time and location) for sources in near-field data (LES or PIV) may be established on this basis.

[1]Supported by AFOSR Grant FA-9550-10-1-0536 and by a Syracuse University Graduate Fellowship.

9:05AM M18.00006 Near field of a transient, acoustically forced transitional and turbulent jets , QI ZHANG, DANIEL BODONY, University of Illinois, Urbana Champaign — Acoustic liners are widely used to reduce aircraft engine noise. They work by converting acoustic-bound energy into vorticity-bound energy, in the form of a transient jet, at an orifice that is very small relative to the incident sound wavelength. At low sound amplitudes (< 130 dB) the forced jet is laminar. At higher amplitudes (>= 150 dB) vortical instabilities appear and the jet becomes turbulent. In this work the behavior of transitional and fully turbulent transient jets are studied using direct numerical simulations of the compressible Navier-Stokes equations. We focus on the near-aperture dynamics of the acoustically-forced fluid by quantifying the jets' phase-averaged properties and linking these to a reduced order dynamical model with the objective of understanding the motion of transient turbulent jets. Results indicate that boundary layer separation from the orifice walls is critical to seeding instabilities within the jets as they develop while at later times disturbances from the previous acoustic cycle reinforce the jets' unsteadiness.

9:18AM M18.00007 Analysis of the near-field of an adjoint-based noise controlled Mach 1.3 turbulent jet , JEONGLAE KIM, DANIEL BODONY, JONATHAN FREUND, University of Illinois at Urbana-Champaign — A noise-reduced Mach 1.3 turbulent jet is investigated to identify its sound-generating mechanisms. The adjoint of the perturbed and linearized Navier–Stokes equations is used in an inverse fashion to provide the control sensitivity to directly reduce sound radiation. The accuracy of this gradient in control parameter space is assessed by examining the variation of the cost functional for small-amplitude controls. The control found by the optimization algorithm reduces noise mainly via suppressing intermittent, large-amplitude acoustic radiation at lower angles and at jet preferred mode frequencies. This observation is similar to the adjoint-based optimization of sound radiated by two-dimensional non-turbulent mixing layers and suggests the subtle modification of large-scale vortical motions for noise reduction. The proper orthogonal decomposition is applied to the near-field pressure fluctuations and the changes in the optimal, orthogonal basis due to the control are analyzed. When controlled, the downstream evolution of large-scale structures shows less abrupt changes, which may be associated with the suppression of the intermittent acoustic radiation. The noise-reducing control is broadbanded in both space and time and demonstrates the characteristics of modulated instability wave packets. Optimizations are ongoing.

9:31AM M18.00008 Large-eddy simulation for the prediction of supersonic rectangular jet noise[1] , JOSEPH W. NICHOLS, FRANK E. HAM, SANJIVA K. LELE, Stanford U., JAMES E. BRIDGES, NASA Glenn Research Center — We investigate the noise from isothermal and heated under-expanded supersonic turbulent jets issuing from a rectangular nozzle of aspect ratio 4:1 using high-fidelity unstructured large-eddy simulation (LES) and acoustic projection based on the Ffowcs-Williams Hawkings (FWH) equations. The nozzle/flow interaction is directly included by simulating the flow in and around the nozzle in addition to the jet plume downstream. A grid resolution study is performed and results are shown for unstructured meshes containing up to 300 million control volumes, generated by a massively parallel code scaled to as many as 65,536 processors. Validated against laboratory measurements using a nozzle of precisely the same geometry, we find that mesh isotropy is a key factor in determining the quality of the far-field aeroacoustic predictions. The full flow fields produced by the simulation, in conjunction with particle image velocimetry (PIV) data measured from experiment, allow for a detailed examination of the interaction of large-scale coherent flow features and the resultant far-field noise, and its subsequent modification in the presence of heating.

[1]Supported by NASA grant NNX07AC94A and PSAAP, with computational resources from a DoD HPCMP CAP-2 project.

9:44AM M18.00009 High Speed Shadowgraph Images around a Mach 1.5 Cavity Flow Field , RYAN SCHMIT, FRANK SEMMELMAYER, MITCHELL HAVERKAMP, JAMES GROVE, Air Force Research Laboratory — An examination of a rectangular cavity with an L/D of 5.67 was tested at Mach 0.7 and 1.5 with corresponding Reynolds numbers of 2x106/ft and 2.3x106/ft, respectively. High speed shadowgraph movies were simultaneously sampled with the dynamic pressure sensors at 75 kHz. Fourier analysis was performed on the high speed movies as well as the dynamic pressure data which resulted in determining the locations of dominant cavity frequencies in the flow field. From the high speed shadowgraph movies, observations of the in the cavity flow physics are discussed. Several cavity related issues are examined e.g. How do vortices form in the shear layer? What is the actual starting mechanism for these cavity acoustic tones? How do the cavity acoustic tones affect the shear layer?

9:57AM M18.00010 Cavity noise control using a small obstacle on cavity floor , SEIICHIRO IZAWA, HIROYUKI NAKASHIMA, YOICHIRO NISHI, MASAYA SHIGETA, YU FUKUNISHI, Tohoku University — An experimental and numerical study to control the cavity noise using a small two-dimensional rectangular cylinder set on a cavity floor has been carried out. In the experiment, it was found that the cavity noise could be reduced to a background noise level by placing a two-dimensional rectangular cylinder at a proper location of the cavity floor. PIV measurement of the flow inside the cavity indicated that flow was forced to separate by the obstacle, changing the recirculating flow inside the cavity. The numerical simulation of the same flow field revealed that the small bump on the cavity floor introduced a large number of longitudinal vortices which interacted with the main shear layer of the cavity flow. The interaction deformed the spanwise vortices, and hence suppressed the cavity noise.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M19 Instability General I 322

8:00AM M19.00001 Spatially localized solutions of plane Couette flow , JOHN GIBSON, EVAN BRAND, University of New Hampshire — We present several new spatially localized solutions of plane Couette flow, each with finite spanwise extent and periodic streamwise structure, and with several different symmetry groups. One new solution is a localized version of the "hairpin vortex" solution of plane Couette flow discovered independently by Itano and Generalis (PRL 2009) and Gibson et al (JFM 2009). The new solutions notably do not exhibit the homoclinic snaking seen in the localized solutions of Schneider et. al (PRL 2010). We also show that the exponential decay rate of the tails of the localized solutions is governed by the wavenumber of the solution's streamwise periodicity.

8:13AM M19.00002 Linear stability of inviscid free surface parallel shear flows[1] , YURIKO RENARDY, MICHAEL RENARDY, Dept of Mathematics, Virginia Tech — We study the linear stability of a class of inviscid parallel shear flows of finite depth with a free surface at the top boundary and a wall at the bottom boundary. We include gravity, but not surface tension. Sample velocity profiles include Poiseuille flow and the hyperbolic tangent shear layer. We determine stability boundaries with respect to the wavenumber of the perturbation, and the relationship of the neutral limiting modes to the velocities at the bottom, at extrema of the baseflow and at inflection points.

[1]Funded by NSF DMS.

8:26AM M19.00003 Velocities of laminar-turbulent fronts in plane Couette flow , YOHANN DUGUET, LIMSI-CNRS, PHILIPP SCHLATTER, DAN HENNINGSON, KTH Stockholm — We investigate numerically the motion of the fronts separating laminar from turbulent zones in subcritical shear flows, with an emphasis on localised turbulent spots in plane Couette flow. A data-driven stochastic analysis of the simulation data is used to compute the average velocities of fronts, in the special cases where they are constrained to travel either in the spanwise or streamwise direction, as the Reynolds number *Re* varies. A quantitative comparison is performed with data for the front velocities of unconstrained spots (from large domain computations). It reveals the dynamical role of a large-scale, nearly two-dimensional secondary flow, which accelerates the front in the spanwise direction and deccelerates it in the streamwise direction. An extrapolation of the underlying linear advection mechanism to the lowest-transitional values of *Re* explains some of the self-organisation properties of laminar-turbulent patterns in the intermittent regime.

8:39AM M19.00004 Experiments on the onset of turbulence in shear flows , KERSTIN AVILA, BJÖRN HOF, Max Planck Institute for Dynamics and Self-Organization — The onset of turbulence in pipe flow occurs via competition of two contrary processes, relaminarization and spreading of turbulence. The timescales of these processes are balanced at Reynolds number $Re = 2040 \pm 10$ and set the onset of sustained turbulence in pipe flow [1]. However, the extremely long timescales of these processes in pipe flow make it impossible to measure important signatures of criticality, such as the scaling of turbulent fraction. This information would be helpful in determining the phase transition class of the onset of turbulence in shear flows. With this goal in mind we have built a Taylor-Couette experiment with an aspect ratio of more than 200 and an azimuthal length of more than 300 gap-widths, allowing us to measure turbulent fractions in the vicinity of the critical point. We also analyze the primary stability of the flow in the counter-rotating regime and present lifetime measurements.

[1] K. Avila, D. Moxey, A. De Lozar, M. Avila, D. Barkley, B. Hof, Science 333(6039), 192-196, 2011

8:52AM M19.00005 Simple models for shear flow transition , DWIGHT BARKLEY, University of Warwick — I will discuss recent developments in modeling transitional shear flows with simple two-variable models. Both pipe flow and plane Couette flow are considered. The essential insight is that most large-scale features of these shear flows can be traced to a change from excitability to bistability in the local dynamics. Models are presented in two variables, turbulence intensity and mean shear. A PDE model of pipe flow captures the essence of the puff-slug transition as a change from excitability to bistability. Extended models with turbulence as deterministic transient chaos or multiplicative noise reproduce almost all large-scale features of transitional pipe flow. In particular they capture metastable localized puffs, puff splitting, slugs, localized edge states, a continuous transition to sustained turbulence via spatiotemporal intermittency (directed percolation), and a subsequent increase in turbulence fraction towards uniform, featureless turbulence. A model that additionally takes into account the symmetries of plane Couette flow reproduces localized turbulence and periodic turbulent-laminar bands.

9:05AM M19.00006 Spatio-temporal dynamics in transitional shear flows , BRUNO ECKHARDT, Philipps-Universitaet Marburg, KORINNA ALLHOFF — The spatio-temporal dynamics of transition pipe flow is dominated by the interplay between a local decay of turbulent puffs and the spreading of turbulence to neighboring regions. Much of the dynamics can be captured by a cellular model with two parameters that describe the local persistence and the spreading. The model shows a transition from a transient turbulence to a persistent one for sufficiently strong spreading. The bulk properties of the model fall into the universality class of 1+1-d directed percolation. The model can also be used to analyze the dynamics of local excitations. These localized excitations show an initial phase of spreading, which is followed by a slower spreading in the parameter range of persistent turbulence, or a contraction and a decay for transient turbulence. The observations are also compared to observations on turbulent spots in plane Couette flow.

9:18AM M19.00007 Wake evolution and trailing vortex instabilities , YLVA ODEMARK, JENS H.M. FRANSSON, KTH Royal Institute of Technology — The production losses and inhomogeneous loads of wind power turbines placed in the wake of another turbine is a well-known problem when building new wind power farms, and a subject of intensive research. The present work aims at developing an increased understanding of the behaviour of turbine wakes, with special regard to wake evolution and the stability of the trailing vortices. Single point velocity measurements with hot-wire anemometry were performed in the wake of a small-scale model turbine. The model was placed in the middle of the wind tunnel test section, outside the boundary layers from the wind tunnel walls. In order to study the stability of the wake and the trailing vortices, a disturbance was introduced at the end of the nacelle. This was accomplished through two orifices perpendicular to the main flow, which were connected to a high-pressure tank and two fast-switching valves. Both varicose and sinusoidal modes of different frequencies could be triggered. By also triggering the measurements on the blade passage, the meandering of the wake and the disturbance frequency, phase averaged results could be computed. The results for different frequencies as well as studies of wake evolution will be presented.

9:31AM M19.00008 The local and global stability of confined planar wakes at Re = 100 ,

MATTHEW JUNIPER, University of Cambridge, OUTI TAMMISOLA, FREDRIK LUNDELL, KTH, Stockholm — At high Reynolds numbers, wake flows become more unstable when they are confined between two flat plates. At Reynolds numbers around 100, however, global stability analyses suggest that such flows become more stable when confined, while local stability analyses suggest that they become more unstable. In this theoretical and numerical study, we combine global and local stability analyses of planar wake flows at Re = 100 to resolve this apparent contradiction. We find that confinement acts in three ways: it modifies the length of the recirculation zone if one exists, it brings the boundary layers closer to the shear layers, and it can make the flow more locally absolutely unstable. In wake flows at Re = 100 with free slip boundaries, the first and third effects work together to make the flow more unstable. In wake flows at Re = 100 with no slip boundaries, the first two effects work against the third to make the flow more stable. By combining local and global analyses, we have been able to isolate these three effects and resolve the apparent contradictions in previous work.

9:44AM M19.00009 Instabilities in the Wake of a Thin Disk with Differents Aspect Ratio ,

TOMASZ BOBINSKI, Warsaw Univ. of Technology, Poland, SOPHIE GOUJON-DURAND, TRISTAN CAMBONIE, JOSE EDUARDO WESFREID, PMMH (ESPCI-CNRS) Paris, France — Flow past a disk was investigated experimentally in a water channel. Systematic experiments with flow visualisation and PIV measurements are presented in order to characterize the flow instabilities. As an extention of previous work, we present results of investigations on disks with different aspect ratio (diameter/thickness) varying from 1 to 24, in the range of the Reynolds numbers from 50 to 500, where stationary and oscillatory instability appear. It is presented the influence of the disk aspect ratio on the evolution of perturbations, the corresponding value of onset instability and the bifurcation branches on the instability. On basis of obtained in PIV measurements vorticity fields it was performed modal analysis. We present results of 3 dimensional, 3 components velocimetry showing detailed information about the physical mechanism of hairpin generation and compare with results in the case of instabilities behind a sphere.

9:57AM M19.00010 Dynamical study of the three dimensional Saffman-Taylor problem ,

MATTEO NICOLI, HERVÉ HENRY, MATHIS PLAPP, Ecole Polytechnique - CNRS — The mathematical generalization of the Saffman-Taylor problem to three spatial dimensions is straightforward but, nevertheless, it has not been widely studied. Recently, Levine and Tu [Phys. Rev. A **45**, 1044 (1992)] solved numerically the problem in the axisymmetric tube geometry finding several solution branches which merge for positive values of the rescaled surface tension parameter $\bar{\gamma}$ (of the order of 10^{-3}). Unlike the two dimensional case, it seems that for this geometry does not exists any axisymmetric solution below this threshold. We have developed a phase-field model of two viscous flows to investigate the dynamics of the 3D Saffman-Taylor problem in the regime of small $\bar{\gamma}$. Full three dimensional simulations in a channel with square section and two dimensional axisymmetric simulations in the tube geometry show that the growing finger undergoes a Plateau-Rayleigh instability leading to pinch-off at the finger tail. Through the linear stability analysis of the tube solution in the axisymmetric geometry, we show that the solutions found by Levine and Tu are unstable for any value of $\bar{\gamma}$. Our phase-field model reproduces accurately this linear prediction and allows to study the influence of the finger tip on the pinch-off velocity. Moreover, we observe that the interface between the two fluids undergoes a tip splitting instability for $\bar{\gamma} < 6.5 \times 10^{-3}$, spoiling the stability of the Saffman-Taylor finger.

Tuesday, November 22, 2011 8:00AM - 10:10AM –

Session M20 Instability in Shear Layers 323

8:00AM M20.00001 Study of Discrete Modes Branching in High-Speed Boundary Layers[1] ,

ANATOLI TUMIN, The University of Arizona, YULI LIFSHITZ, DAVID DEGANI, Technion-Israel Institute of Technology — The branching of discrete modes in high-speed boundary layers is investigated using the Parabolized Stability Equations (PSE). The fast and slow discrete modes associated with the fast and slow acoustic modes, respectively, are considered in high-speed boundary layers over adiabatic and cooled walls. Whereas the conventional Linear Stability Theory approach leads to singular behavior in the vicinity of the fast mode synchronization with the entropy and vorticity modes, the PSE results do not reveal singular behavior of the solution and are consistent with the available Direct Numerical Simulations of perturbations in high-speed boundary layers. Also, the PSE results do not reveal a singular behavior at the point of synchronism of the slow and fast discrete modes.

[1]A. Tumin acknowledges support from the Air Force Office of Scientific research (AFOSR) /NASA/ National Center for Hypersonic Research in Laminar-Turbulent Transition.

8:13AM M20.00002 Enhancement of thermal fluctuations in Plane Couette Flow[1] ,

JOSE M. ORTIZ DE ZARATE, Complutense University, JAN V. SENGERS, University of Maryland — Mode-coupling phenomena in systems outside equilibrium generically cause an enhancement of thermal fluctuations. These enhancements can be studied by Landau's fluctuating hydrodynamics. Here we present a detailed study for the case of plane Couette flow based on stochastically forced Orr-Sommerfeld and Squire equations. The forcing arises from random contributions to the stress tensor due to the stochastic nature of molecular collisions. This intrinsic stochastic forcing is then amplified by mode- coupling mechanisms associated with the shear flow. We discuss the different coupling mechanisms, the most important one being the direct coupling between fluctuations of the wall-normal velocity and vorticity. The most pronounced effect is amplification of wall-normal vorticity fluctuations with a spanwise modulation at dimensionless wave numbers $q_{\|}$ around 1.5.

[1]Financial support: MICINN FIS2008/03801

8:26AM M20.00003 Vortex-wave interactions/self-sustained processes in shear layers ,

PHILIP HALL, Imperial College — It was shown by Hall and Sherwin (2010) that the so-called lower branch self-sustained processes which have been found in the last decade are finite Reynolds numbers versions of the vortex-wave interactions described in a number of papers in the early 1990's by Hall and Smith. Here the corresponding structures are developed in natural convection and a much simplified set of interaction equations are derived. The states are shown to be subharmonic with respect to the spanwise variable and their stability is discussed. The relevance of these states to Couette flow is discussed.

8:39AM M20.00004 Optimal disturbances in shearing and swirling flows[1] ,

CONOR DALY, University of Cambridge — Over the past twenty years transient energy density growth of linearly stable disturbances has shown to be the likely instigator for transition to turbulence in parallel shear flows. In this vein, optimal linear perturbations are calculated for two flows which have a mixture of forces acting on the fluid body. These are; rotating plane Couette flow (RPCF), which combines pressure-driven shear and swirl, and cylindrical Couette-Poiseuille flow (CCPF), which combines pressure-driven and Couette shear. Contours are presented of the maximum achievable linear transient growth, G, over the full range of wavenumbers within the linearly stable parameter regimes. Reference is made to experimental works on each flow and we examine the role that optimal disturbances have in the different transition phenomena that are observed. It is found that the contours of G fall qualitatively alongside the points of transition in the two flows, in support of the notion that large linear transient growth can act a precursor to transition. Despite the combination of effects acting on each fluid, transition in both flows falls in the range $10^2 < G < 10^{2.5}$ suggesting that in both flows the same mechanism may be at work.

[1]This work is funded by EPSRC.

8:52AM M20.00005 Transversal motion and flow structure of fully nonlinear streaks in a laminar boundary layer , JUAN ANGEL MARTIN, CARLOS MARTEL, Universidad Politecnica de Madrid, DENLIA TEAM — Typical streak computations present in the literature correspond to linear streaks or to small amplitude nonlinear streaks computed using DNS or nonlinear PSE. We use the Reduced Navier-Stokes (RNS) equations to compute the streamwise evolution of fully non-linear streaks with high amplitude in a laminar flat plate boundary layer. The RNS formulation provides Reynolds number independent solutions that are asymptotically exact in the limit $Re \gg 1$, it requires much less computational effort than DNS, and it does not have the consistency and convergence problems of the PSE. We present various streak computations to show that the flow configuration changes substantially when the amplitude of the streaks grows and the nonlinear effects come into play. The transversal motion (in the wall normal-streamwise plane) becomes more important and strongly distorts the streamwise velocity profiles, that end up being quite different from those of the linear case. We analyze in detail the resulting flow patterns for the nonlinearly saturated streaks and compare them with available experimental results.

9:05AM M20.00006 Numerical study of instability in a laminar plane wall jet , LARS-UVE SCHRADER, CATHERINE MAVRIPLIS, University of Ottawa, Canada, LUCA BRANDT, KTH Mechanics, Stockholm, Sweden, UNIVERSITY OF OTTAWA, CANADA COLLABORATION, KTH MECHANICS, STOCKHOLM, SWEDEN COLLABORATION — A two-dimensional jet released parallel to a wall is called a plane wall jet. This flow type plays an important role in cooling of surfaces, e.g. in laptop processors. Developed laminar wall jets are nearly self-similar in the streamwise direction. Their instability was investigated by linear stability theory (LST) in previous studies. In this presentation, we report a direct numerical simulation (DNS) study of the linear spatial evolution of the wall jet instabilities and compare our DNS results with findings from LST. We find large discrepancies of the linear disturbance growth rates and attribute this to the incompatibility of the parallel flow assumption of LST with the rapidly evolving plane wall jet. A correction to the instability map is suggested, where especially the "stable hole" (a patch of stable conditions within the unstable region) turns out to be significantly larger than known so far. We also computed optimal initial disturbances using a method based on the adjoint Navier-Stokes equations. These disturbances evolve into traveling wave packets with maximum possible disturbance kinetic energy.

9:18AM M20.00007 Non-linear single-wave Kelvin-Helmholtz instability in a channel[1] , ANNA-GRAZIA ORAZZO, Universita di Napoli Federico II, Dept. of Aerospace Engineering DIAS, Italy, GENNARO COPPOLA, Universita di Napoli Federico II, Dept. of Energetics and Applied Thermo-fluid-dynamics DETEC, Italy, LUIGI DE LUCA, Universita di Napoli Federico II, Dept. of Aerospace Engineering DIAS, Italy — A stratified viscous gas-liquid two-phase flow confined in a horizontal channel is studied, surface tension effects being included. Contrary to previous papers of literature, where a parallel flow configuration is classically analyzed with plug-velocity profile in both fluids, here the flow is spatially developing starting from a plug-plug profile at the channel entrance. The sudden change of interface boundary condition produces the flow development and the emergence of a solitary Kelvin-Helmholtz wave, whose formation and evolution, inherently non linear, are studied through numerical simulations based on the Volume of Fluid (VOF) technique. The amplitude growth rate and the propagation velocity of the wave at early instants agree closely with the predictions of a straightforward model. Later times simulations show the wave break-up in small droplets.

[1]This work was supported by AVIO S.p.A., Turin, Italy.

9:31AM M20.00008 A nonlinear variational approach to triggering transition in plane Couette flow , S.M.E. RABIN, DAMTP, University of Cambridge, C.P. CAULFIELD, BPI & DAMTP, University of Cambridge, R.R. KERSWELL, University of Bristol — The study of the stability of shear flows has a long history dating back more than a hundred years. Understanding how and when turbulence emerges in such flows is highly significant to many processes studied throughout engineering. A feature of turbulent flows are that they have significantly higher kinetic energy than those that remain laminar. As a consequence research has focused on optimizing kinetic energy at a specific target time, initially for the linearized Navier Stokes equations (Butler & Farrell 1992), and more recently for the full Navier Stokes equations (Pringle & Kerwell 2010, Cherubini et al 2010). The belief is that by achieving high energies turbulence can be triggered. An alternative theory is that optimizing time averaged dissipation is more effective at triggering turbulence (Monokrousos et al 2011). In this study we optimize kinetic energy growth over all initial states and all target times for a given initial kinetic energy in order to reveal the "minimal seed" for turbulence (the disturbance of lowest kinetic energy which can trigger turbulence). We present results for a geometry originally considered by Butler & Farrell (1992) and also compare our minimal seed prediction with that made recently by Monokrousos et al (2011).

9:44AM M20.00009 Dynamics of vorticity defects in layered stratified shear flows , C.P. CAULFIELD, BPI/DAMTP, U. of Cambridge, A. ROY, Engineering Mechanics Unit, JNCASR, Bangalore, N.J. BALMFORTH, Mathematics/EOS, U. of British Columbia — Layered stratified flows, where relatively deep regions of weak stratification are separated by thinner interfacial layers of substantially stronger density gradient are commonly observed in nature. If such flows are subjected to vertical shear, it is well-known that a wide range of qualitatively different instabilities may develop. For example, the three-layer, two interface case is susceptible to a "Taylor" instability which, although superficially similar to the classic Kelvin-Helmholtz instability, is actually qualitatively different in its growth mechanism. The investigation of the nonlinear dynamics of this instability, and to a lesser extent the single-interface "Holmboe" instability, has proved difficult, as the need to resolve the associated sharp density gradients places heavy demands on the required numerical resolutions for simulation. However, we show that it is possible to gain insight into the key nonlinear dynamics of such layered stratified shear flows by generalizing a reduced matched asymptotic "vorticity defect" model (N. J. Balmforth et al. *J. Fluid Mech.* **333**, 197 [1997]) to include the dynamical effects of density variations. We particularly focus on investigating the finite amplitude structure of the saturated primary Taylor instability, and the properties of the secondary instabilities to which Taylor and Holmboe instabilities are susceptible.

9:57AM M20.00010 Laminar streak enhancement using streamwise grooves , CARLOS MARTEL, JUAN ÁNGEL MARTÍN, Universidad Politécnica de Madrid — Laminar streak promotion in a flat plate boundary layer results in an increase of the stability of the Tollmien-Schlichting waves with respect to that of the 2D Blasius profile. This stabilization delays the laminar-turbulent transition, increasing the laminar phase of the flow. The stabilization effect is stronger for higher streak amplitudes, and therefore simple ways of generating high amplitude stable streaks are sought to be used as boundary layer flow control methods. In a recent experiment [Tallamelli & Franson,AIAA 2010-4291] high amplitude stable steady streaks have been produced using Miniature Vortex Generators (MGVs), where one array of MGVs is used to excite the streak and a second array is used downstream to enhance their amplitude. In this presentation we numerically explore the possibility of enhancing the streaks using a different passive mechanism: streamwise grooves carved in the plate. We will present some numerical simulations for different values of the spanwise period of the streaks and of the grooves, and we will show the combinations that provide maximum streak amplitude.

Tuesday, November 22, 2011 8:00AM - 9:57AM –
Session M21 Vortex Dynamics VI 324-325

8:00AM M21.00001 Vortex Ring Properties Before and After Interaction with a Screen[1] , JOHN HRYNUK, DOUGLAS BOHL, Clarkson University — The interaction of a vortex ring with a screen was investigated using Molecular Tagging Velocimetry (MTV). The screen had a porosity of 65% and a small wire diameter (D_w=0.0178 cm) with respect to the vortex ring diameter (D_w/ D_{ring}=5.23x10^{-3}). The results show that as the vortex ring approached the screen the initial interaction was much like that for a vortex ring/solid wall interaction. Specifically, the vortex ring slowed down and expanded as it approached the screen. A secondary vortex was formed that separated from the wall and orbited the primary vortex. Because the wall was porous the primary vortex ring passed through the wall, however, the secondary ring slowly convected back upstream. Data showed that the peak vorticity of the primary vortex dropped by nearly a factor of 4 after passing through while the radius increased in size only slightly (15%). The circulation of the primary vortex before and after the interaction decreased by more than a factor of 2 confirming conclusions from prior flow visualization results. The secondary vortex was both smaller and weaker than the primary vortex as is observed for vortex/wall interactions.

[1]Research Supported by NSF Grant 0845882.

8:13AM M21.00002 ABSTRACT MOVED TO L14.00009 —

8:26AM M21.00003 Vortex interaction during the pinch-off process in a starting jet , LEI GAO — Experiments on starting jets at three Reynolds number (Re = 2600, 4100, 5600) were performed to investigate the effect of interaction of the leading vortex ring (LVR) with trailing secondary vortices on the pinch-off process. The velocity and vorticity fields of the starting jet of were obtained by using particle image velocimetry (PIV). It was found that after the formation number (i.e., the onset of the pinch-off process), secondary vortices develop in the trailing jet owing to the shear layer instability. These vortices then affect the dynamics of the trailing jet, especially the vorticity transportation into LVR. In the cases at higher Reynolds number, merging of LVR with the secondary vortices occurs during the pinch-off process, whose completion is indicative of the physical separation of LVR from its trailing jet. Therefore, although the formation number is relatively universal in all tested cases, the time of separation (i.e., the end of the pinch-off process) is actually sensitive to the Reynolds number due to its effect on the shear layer instability.

8:39AM M21.00004 Interaction between leading and trailing edge vortex shedding: effects of bluff body geometry , ZACHARY TAYLOR, GREGORY KOPP, UWO, ROI GURKA, BGU — Elongated bluff bodies are distinguished from shorter bluff bodies (e.g., circular cylinders) by the fact that they have separating-reattaching flow at the leading edge as well as having vortex shedding at the trailing edge. Engineering examples of these bodies include heat exchanger fins and long-span suspension bridges. We have performed experiments on elongated bluff bodies of varying geometry. These experiments have been performed at Reynolds numbers O(10^4) based on the thickness of the model. Both surface pressure measurements (using 512 simultaneously sampled pressure taps) and PIV are used to quantify the flow fields of these bodies. The leading edge separation angle is controlled by changing the leading edge geometry. It is observed that the size of the leading edge separation bubble increases with increasing leading edge separation angle. As the size of the leading edge separation bubble increases, it is shown to continually decrease the shedding frequency for a given elongation ratio. It is suggested that the shedding frequency is diminished because the trailing edge vortex shedding is affected by the structures being shed from the leading edge separation bubble. The implications of this competition between leading and trailing edge flows will be explored.

8:52AM M21.00005 ABSTRACT WITHDRAWN —

9:05AM M21.00006 The formation of vortex rings from elliptical nozzles , CLARA O'FARRELL, ROBERT WHITTLESEY, JOHN O. DABIRI, California Institute of Technology — It is known that there is 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.[1] This transition, termed "vortex pinch-off," is predicted by an energy-maximization argument due to Kelvin and Benjamin.[2] However, the Kelvin-Benjamin principle does not apply to non-axisymmetric flows, and the dynamics of the formation of non-axisymmetric vortex rings remain largely unknown. We consider the formation of vortex rings from elliptical nozzles, and compare it to that of axisymmetric vortex rings. By performing PIV on several planes along the perimeter of the elliptical nozzle, we study the effect of varying curvature on vortex formation.

[1]Gharib *et al.*, J. Fluid Mech., **360**, p. 121, 1998.
[2]Benjamin, In *Applications of Methods of Functional Analysis to Problems in Mechanics*, Springer, 1976.

9:18AM M21.00007 Early pinch-off in formation of consecutive vortex rings due to vortex interaction , JIFENG PENG, University of Alaska Fairbanks — The formation of an isolated vortex ring from a starting jet is a limited process described by the non-dimensional formation time. During formation, the vorticity flux of the jet shear layer is entrained into the forming ring until the formation time reaches a limit, upon which the ring pinches off from the trailing jet (Gharib et al. 1998). The limiting formation time can be attributed to the Kevin-Benjamin principle, which dictates that pinch-off occurs when the shear layer is no longer able to deliver the energy required for the existence of a steady vortex ring. In formation of consecutive vortex rings from a pulsed jet, due to interaction between vortex rings, the limiting ring growth process depends not only on the formation time, but also on the pulsing frequency of the jet. This experimental study on a classic piston-cylinder arrangement finds that when pulsing frequency is high and interaction between rings is strong, the forming ring pinches off at a significantly smaller formation time compared with that in isolated ring formation. A theoretical model is developed to explain the reduced limiting formation time in consecutive ring formation.

9:31AM M21.00008 Improving propulsive efficiency through passive mechanisms using a Starling vortex generator[1] , ROBERT WHITTLESEY, JOHN DABIRI, California Institute of Technology — Ruiz et al. (2011) demonstrated that pulsed propulsion with vortex rings, much like those seen in the wake of jellyfish and squid, can greatly enhance the overall efficiency of submersible vehicles. The objective of the present research is to achieve pulsed propulsion passively using a Starling vortex generator which consists of a collapsible tube within an airtight box. Recent work has shown that a Starling vortex generator is able to generate vortex rings, which indicates enhanced propulsion, while requiring less energy to generate pulsatility than the system by Ruiz et al. (2011). Current work is focused on conducting an experimental parameter study to determine an empirical scaling law suitable for design purposes, with the aim to integrate the device into a full-scale unmanned undersea vehicle.

[1]Support is greatly appreciated from ONR Awards N000140810918 and N000141010137.

9:44AM M21.00009 Experimental study of a vortex ring impacting a smart material-based cantilevered plate , SEAN PETERSON, University of Waterloo, MAURIZIO PORFIRI, Polytechnic Institute of New York University — Recent developments in lightweight smart materials have generated scientific and technological advancements in small scale energy harvesting for powering low-consumption electronic devices. Often, energy is harvested from base excitation of a cantilevered smart material strip. In this case, the encompassing fluid acts as a passive damper, reducing the vibration amplitude and frequency, which reduces the harvesting capacity. By comparison, relatively few research efforts to date have explored the feasibility of using smart materials for harvesting energy directly from fluid motion. In this paper we employ vortex rings as the source from which to extract energy and use an ionic polymer metal composite (IPMC) strip in a cantilevered configuration as the harvesting device. Vortex rings, generated using a piston/cylinder arrangement submersed in water, are fired at the IPMC harvester and the resulting impact is recorded using a high speed video camera. The vortex ring propagation and circulation are estimated using flow visualization and particle image velocimetry. The plate deflection and electrical output are recorded as functions of time and correlated to the vortex strength and geometry.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M23 Flow Control V: Actuators II 326

8:00AM M23.00001 Supersonic Jet Noise Reduction , EPHRAIM GUTMARK, NICK HEEB, University of Cincinnati, JUNHUI LIU, KAILAS KAILASANATH, U.S. Naval Research Laboratory — Three noise reduction technologies have been examined experimentally as they have been applied to overexpanded, perfectly expanded and underexpanded supersonic jets from convergent-divergent nozzles. The technologies include chevrons, fluidic injection and fluidically enhanced chevrons. The flowfield was measured by shadowgraph and particle image velocimetry (PIV). The acoustics were measured by near and far-field microphone. Chevrons were shown to reduce or eliminate screech, reduce broadband shock associated noise and reduce mixing noise. Fluidic injection was shown to reduce screech, reduce broadband shock associated noise and mixing noise. It also shifts the shock-associated noise peaks to higher frequency and generates increased high frequency noise as chevrons do. The fluidic injection produces the same reduction near x/D = 10 at mid frequencies and the same increase in high frequencies near the nozzle as chevrons. Both noise reduction techniques reduce the size of the large scale structures and so both reduce BBSN by the same mechanism. The principal difference between chevrons and fluidic injection is that for constant injection mass flow the effectiveness of fluidic injection increases with decreasing values of M_j while for chevrons the trend is reversed.

8:13AM M23.00002 Towards Feedback Control of Bypass Transition Using Plasma Actuators[1] , RONALD HANSON, PHILIPPE LAVOIE, University of Toronto, Institute for Aerospace Studies, KYLE BADE, AHMED NAGUIB, Michigan State University, Department of Mechanical Engineering — Feedback flow control is applied to the transient-growth instability occurring in a Blasius boundary layer for the purpose of delaying bypass transition. The control signal is based on empirical modelling of the input/output flow response. The latter is obtained for both the main disturbance, generated by a roughness element array, as well as the control disturbance, introduced using a spanwise array of plasma actuators. Specifically, the targeted disturbance is characterized with streamwise velocity measurements in cross-flow planes in conjunction with streamwise-wall-shear-stress measurements over two fundamental spanwise disturbance wavelengths, downstream of the actuator location. Correlations between spanwise profiles of the streamwise strear stress and the measured disturbance velocity profiles are used to specify the control signal supplied to the actuators in conjunction with input/output model of the flow response to plasma actuation. The aim of this study is to minimize the residual disturbance energy in a closed-loop framework.

[1]NSERC and NSF grant number: CMMI 0932546

8:26AM M23.00003 The effect of orifice shape on synthetic jet efficiency , BARTON SMITH, DAVID NANI, Utah State University Mechanical and Aerospace Engineering — Many studies have demonstrated that synthetic jets can be sensitive to the cavity or orifice shape. In most cases, the performance of the jet with different configurations is compared for fixed driver input voltage or fixed driver displacement. Neither of these quantities accurately reflect the efficiency of the actuator. A sharp inside edge of a synthetic jet orifice can result in separated flow and increased momentum flux (due to the decreased flow area) for a fixed driver displacement. This can lead one to believe that efficiency has been improved, when, in fact, much more power was required for the driver. Similarly, altering the driving waveform from a sinusoid has been reported to improve efficiency. Acoustic power, which is the time-average of volume flow rate through the orifice multiplied by the driving pressure, accurately accounts for the amount of power required to drive the actuator. In this study, we study the efficiency of a round synthetic jet actuator as a function of the radius of the inside of the orifice. Simultaneous PIV at the jet exit and cavity pressure measurements are used to compute the acoustic power required to drive the actuator. The resultant momentum flux of the jet is used as a measure of strength of the jet. Results are obtained for a range of Reynolds numbers and displacement amplitudes. Not surprisingly, it is found that rounding the inside of the orifice improves the efficiency of a synthetic jet.

8:39AM M23.00004 A Parametric Study of Plasma Vortex Generators for Active Flow Control , MICHAEL WICKS, ERAS NOEL, FLINT THOMAS, THOMAS CORKE, University of Notre Dame — The performance of plasma streamwise vortex generators (PSVG) based on a dielectric barrier discharge is characterized experimentally. A PSVG array is flush mounted on a turbulent boundary layer development plate, which allows control of the flow conditions upstream. The performance of the PSVG is characterized by nonintrusive flow field measurements utilizing both LDV and PIV. The primary metric for characterizing the performance of the PSVG is the magnitude of streamwise vorticity produced. Through a series of experiments the influence of applied voltage, length of surface electrode, inter-electrode spacing, electrode geometry, yaw angle to the oncoming flow and Reynolds number on PSVG performance is documented. The results are compared to passive vortex generators under comparable flow conditions.

8:52AM M23.00005 Flow-field characterization of DBD plasma actuators as discrete roughness elements for laminar flow control[1] , S.A. CRAIG, R.A. HUMBLE, J.W. HOFFERTH, W.S. SARIC, Texas A&M University — For many years there has been an evolving interest in controlling boundary layer transition on swept-wings. With an appropriate distribution of spanwise-periodic discrete roughness elements (DRE), subcritical wavelengths can be excited which supersede the growth of the most-amplified wavelength, thereby delaying the crossflow-dominated laminar-turbulent transition. To elucidate the physics of annular DBD plasma actuators for potential use as DRE and facilitate effective design, they are studied under quiescent flow conditions using particle image velocimetry a photomultiplier tube (PMT) and a high-speed camera. A complex flow-field is generated by a single aperture that describes a three-dimensional torus accompanied by a downward, wall-normal jet region. The flow-field is sensitive to aperture size and applied voltage. For arrayed actuators, the tori contract dramatically due to the interaction with vortices from adjacent apertures. A PMT in conjunction with a high-speed camera were used to observe the light intensity from the bulk plasma at high temporal resolution and visualize the individual discharge events.

[1]Supported by the NASA/AFOSR National Center for Hypersonic Laminar-Turbulent Transition Research.

9:05AM M23.00006 Active Control of Jet Noise Using High Resolution TRPIV Part 1: POD Analysis , ZACHARY BERGER, Syracuse University , STANISLAV KOSTKA, Spectral Energies , KERWIN LOW, MATT BERRY, Syracuse University , SIVARAM GOGINENI, Spectral Energies, MARK N. GLAUSER, Syracuse University — In the current investigation, we seek to develop advanced flow control techniques for a high speed compressible jet, of nozzle diameter 2". Using hydrodynamic pressure and 10kHz time-resolved PIV in the near-field, as well as acoustic pressure in the far-field, we implement various techniques to further our understanding of this complex flow field. This work focuses on the near-field velocity measurements for the Mach 0.6 and 0.85 jet. The current 2-component PIV setup allows analysis of the streamwise (r-z) plane of the jet with a 1.5D window. The area of interest is x/D = 3-6 (region of the collapse of the potential core), where there are many potential noise sources propagating to the far-field. In-depth POD analyses will be performed in order to correlate the time-dependent POD coefficients with the pressure signals and develop advanced strategies for closed-loop flow control. For the Mach 0.6 case, at a streamwise location of x/D = 3-4.5, a favorable convergence rate of the spatial eigenvalues can be seen from 1000 snapshots; 45% energy from 20 modes.

9:18AM M23.00007 Active Control of Jet Noise Using High Resolution TRPIV Part 2: Velocity-Pressure-Acoustic Correlations , KERWIN LOW, Syracuse University, STANISLAV KOSTKA, Spectral Energies, LLC, ZACHARY BERGER, Syraucse University, MATTHEW BERRY, Syracuse University, SIVARAM GOGINENI, Spectral Energies, LLC, MARK GLAUSER, Syracuse University — We investigate the pressure, velocity and acoustic field of a transonic jet. Test conditions comprise a 2 inch nozzle, analyzing two flow speeds, Mach 0.6 and 0.85, with open loop control explored for the Mach 0.6 case. We make simultaneous measurements of the near-field pressure and far-field acoustics at 40 kHz, alongside 10 kHz time resolved PIV measurements in the r-z plane. Cross correlations are performed exploring how both the near-field Fourier filtered pressure and low dimensional POD modes relate to the far-field acoustics. Of interest are those signatures witch exhibit the strongest correlation with far-field, and subsequently how these structures can be controlled. The goal is to investigate how flow-induced perturbations, via synthetic jet actuators, of the developing shear layer might bring insight into how one may alter the flow such that the far-field acoustic signature is mitigated. The TR-PIV measurements will prove to be a powerful tool in being able to track the propagation of physical structures for both the controlled and uncontrolled jet.

9:31AM M23.00008 Active Flow Control on a Low Aspect Ratio Finite Cylinder , EDWARD DEMAURO, CHIA MIN LEONG, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Oftentimes, flow control about bluff bodies is investigated using quasi-2D shapes that ignore end effects, whereas real world objects are finite and thus exhibit significant three-dimensional flow fields. Therefore, synthetic-jets-based active flow control was studied on a finite cylinder of low aspect ratio (AR = 3), which incorporates large-scale end effects that must be taken into consideration. Surface pressure measurements indicated that the flow field was significantly modified by the activation of the synthetic jet actuators, when compared to the unforced case. Even with a small momentum input, the synthetic jets induce a large spanwise effect (i.e., along the cylinder span). This large scale alteration of the flow field was confirmed visually using time-averaged stereoscopic PIV measurements in the near wake, showing significant wake narrowing and vectoring, along with changes to vorticity concentrations and turbulent quantities.

9:44AM M23.00009 Separation Control on Generic ROBIN Rotorcraft Fuselage Using Plasma Actuators , DUSTIN COLEMAN, University of Notre Dame — Active flow control, in the form of dielectric barrier discharge (DBD) plasma actuators, is applied to a NASA ROBIN mod7 generic rotorcraft fuselage model. The control objective is reduce the massive 3-D flow separation occurring over the aft ramp section of the fuselage, thereby improving the vehicle flight characteristics. The plasma actuation methods investigated include: plasma streamwise vortex generators (PSVGs), as well as steady and unsteady spanwise actuation, combined with passive geometric modifications to the ramp section. Experiments were conducted at freestream Mach and Reynolds numbers of $M_\infty = 0.12$ and $\mathrm{Re}_L = 2.65 \times 10^6$, respectively. Aerodynamic loads from each technique were quantified by means of 3-component force balance measurements (drag, lift, and pitching moment), a 128 count static pressure array, and time-resolved PIV wake surveys. Results are compared with previous studies that utilized active flow control in the form of pulsed jets and combustion actuators.

9:57AM M23.00010 ABSTRACT WITHDRAWN –

Tuesday, November 22, 2011 8:00AM - 9:57AM –
Session M24 Convection and Buoyancy Driven Flows II 327

8:00AM M24.00001 Is horizontal convection really "non-turbulent"? , ALBERTO SCOTTI, Dept. Marine Sciences, UNC-CH, BRIAN WHITE, Dept. of Marine Sciences — The oceanic Meridional Overturning Circulation (MOC) is the slow residual motion that turns over the world ocean over a millennial time scale. What drives the MOC has been the subject of intense speculation. Equatorial heating and polar cooling, i.e. horizontal convection (HC), would seem to be a natural candidate to drive (at least partially) the MOC, but within the oceanographic community HC has been considered irrelevant or nearly so, based primarily on an inference based on a century old experiment by Sandström (1908), though all modern experiments contradict it (Hughes and Griffiths, 2008), and on a theoretical argument that would prevent HC to sustain a true turbulent flow (Paparella and Young, 2002), the latter deemed necessary to achieve mixing. We revisit Paparella and Young's argument with the aid of DNS of HC at Rayleigh number up to 10^{10}. We argue that the latter argument is overly restrictive. On the contrary, analysis of invariants (such as Q-R plots) show that HC possesses the characteristic of turbulent flows. The surprising result is that HC can transport very large quantities of heat and sustain large amount of diapycnal mixing with a surprisingly small amount of dissipation. Hence, HC alone could be an important component behind the MOC, even though it cannot explain the levels of dissipation observed in the ocean, which have to be accounted from other sources.

8:13AM M24.00002 Horizontal convection PIV study , STEFANO DISCETTI, TOMMASO ASTARITA, University of Naples Federico II — Horizontal convection has received limited attention, despite of its well recognized implication in geophysical flows of relevant interest. E.g., oceans are often modeled as being cooled and heated in a thin surface layer, which is nearly horizontal, due to solar irradiance forcing differentially in latitude and determining strong stable density stratification. The phenomenology is simulated in a plexiglass box, with a piecewise thermal boundary condition on its bottom surface while all the other boundaries are thermally insulated. A Particle Image Velocimetry (PIV) investigation of the natural convection generated by a differential thermal boundary condition along the same horizontal boundary is proposed. A preliminary investigation of the structure of the circulation cell, generated by the hot water rising up in correspondence of the heat flux input zone, and sinking down nearby the cooled zone, is carried out. A strong convection cell, consisting of a rising intense plume (filling the whole box height) and a slow downward average flow, are observed in all the tested configurations.

8:26AM M24.00003 Horizontal Convection in the Presence of a Surface Stress[1] , KATARZYNA MATUSIK, UCSD — Horizontal convection, the flow driven by buoyancy fluxes imposed along a horizontal boundary of a fluid, is a model of the meridional overturning circulation of the oceans. We explore features of horizontal convection in the presence of a surface stress. The buoyancy flux is achieved by injecting a plume of dense water into a fresh-water tank, while continuously imposing a fresh-water surface boundary condition. The magnitude of the stress is varied by adjusting the flow rate of fresh water traversing along the surface. We measure the steady-state density field using the Synthetic Schlieren method, and qualitatively observe the flow via discrete dye injections. The vertical and horizontal velocities are determined by PIV techniques. Our goal is to examine the effects of a surface stress of varying magnitude and direction on the general horizontal convection. We compare laboratory data to scaling analyses for the boundary- layer thickness and the strength of the overturning circulation. The addition of a surface stress to horizontal convection may offer insight into the effects of wind on the ocean surface, namely the implications of a kinetic energy source on the overall energetics of the circulation.

[1]NSF Grant

8:39AM M24.00004 DNS of unstable stratification in turbulent channel flow , ALFREDO SOLDATI, University of Udine, FRANCESCO ZONTA, University of Torino — Turbulence subject to unstable stratification (bottom-up heating) is of great interest due to its dynamical importance in engineering and geophysical flows. Unstably-stratified flows often occur in the presence of high temperature gradients. In this situation, it is important to account for variation of the fluid properties with temperature. According to this idea, we use direct numerical simulation to analyze the physics of unstably-stratified turbulence with temperature-dependent fluid properties: specifically, viscosity and thermal expansion coefficient. Due to unstable stratification, wall-normal transport of momentum and heat increases with respect to the neutrally-buoyant case. Results show that flow modifications due to temperature-dependent properties may become important. In particular, the effect of temperature-dependent thermal expansion coefficient is to increase the wall normal transport of momentum and heat. By contrast, the effect of temperature-dependent viscosity appears negligible.

8:52AM M24.00005 Stable stratification in turbulent channel flow , FRANCESCO ZONTA, MIGUEL ONORATO, University of Torino, ALFREDO SOLDATI, University of Udine — Stable stratification (bottom-up cooling) is of great importance, since it is encountered in industrial applications, environmental processes and geophysical flows. Turbulent entrainment and mixing across density interfaces in terrestrial water bodies (oceans, lakes and rivers) or at the boundaries between water bodies and the atmosphere, are just some important examples of stably-stratified flows. In this work we use direct numerical simulation to investigate the fundamental physics of stably-stratified turbulent channel flow of water with temperature dependent viscosity and thermal expansion coefficient. Compared to the neutrally-buoyant case, stable stratification induces a general suppression of turbulence levels, momentum and buoyancy fluxes. We observe that the effect of temperature-dependent fluid properties may be important. The most striking feature produced by temperature-dependent fluid properties is local flow laminarization in the cold side of the domain (for temperature-dependent viscosity) or in the hot side of the domain (for temperature-dependent thermal expansion coefficient).

9:05AM M24.00006 ABSTRACT WITHDRAWN —

9:18AM M24.00007 Unsteady Heat Transfer in Channel Flow using Small-Scale Vorticity Concentrations Effected by a Vibrating Reed , PABLO HIDALGO, ARI GLEZER, Georgia Institute of Technology — Heat transfer enhancement by small-scale vorticity concentrations that are induced within the core flow of a mm-scale heated channel are investigated experimentally. These small-scale motions are engendered by the cross stream vibrations of a streamwise cantilevered reed that spans most of the channel's width. The interactions between the reed the core flow over a range of flow rates lead to the formation, shedding, and advection of time-periodic vorticity concentrations that interact with the wall boundary layers, and increase cross stream mixing of the core flow. Heating of the channel walls is controlled using microfabricated serpentine resistive heaters embedded with streamwise arrays of temperature sensors. It is shown that the actuation disrupts the thermal boundary layers and result in significant enhancement of the local and global heat transfer along the channel compared to the baseline flow in the absence of the reed. The effect of the reed on the cross flow is measured using high resolution particle image velocimetry (PIV), and the reed motion is characterized using a laser-based position sensor. The blockage induced by the presence of the reed and its cross stream motion is characterized using detailed streamwise pressure distributions. Supported by DARPA and UTRC.

9:31AM M24.00008 Mixing from diffusion and natural convection in binary non-equilibrium fluid phases , LAURENCE RONGY, Universite Libre de Bruxelles ULB, KJETIL HAUGEN, ABBAS FIROOZABADI, Yale University — The mixing of two non-equilibrium fluid phases is relevant to a large number of problems in industry and in nature. Important applications are improved oil recovery and carbon sequestration. Our work provides a realistic description of the mixing in non-ideal fluids, which is needed to assess the efficiency of oil extraction and the storage capacity in geological formations. We also show how to determine diffusion coefficients accurately in non-ideal fluids at reservoir conditions from measurements such as pressure evolution and gas-liquid level data whereas most techniques are restricted to measurements at atmospheric pressure only.

9:44AM M24.00009 Buoyancy-driven instabilities of acid-base fronts: the case of a color indicator , L.A. RIOLFO, NLPC, Universite Libre de Bruxelles, Belgium, S. KUSTER, GMP, Universidad de Buenos Aires, Argentina, P.M.J. TREVELYAN, NLPC, Universite Libre de Bruxelles, Belgium, C. EL HASI, A. ZALTS, UNGS, Argentina, C. ALMARCHA, NLPC, Universite Libre de Bruxelles, Belgium, A. D'ONOFRIO, GMP, Universidad de Buenos Aires, Argentina, A. DE WIT, NLPC, Universite Libre de Bruxelles, Belgium — Buoyancy-driven hydrodynamic instabilities of acid-base fronts are studied both experimentally and theoretically in the case where an aqueous solution of a strong acid is put above a denser aqueous solution of a color indicator in the gravity field. The neutralization reaction between the acid and the color indicator as well as their differential diffusion modifies the initially stable density profile in the system and can trigger convective motion both above and below the initial contact line. The type of patterns observed as well as their wavelength and the speed of the reaction front are shown to depend on the value of the initial concentrations of the acid and of the color indicator and on their ratio. A reaction-diffusion model explains how the hydrodynamic instability scenarios change when the concentration of the reactants are varied.

Tuesday, November 22, 2011 8:00AM - 10:10AM —

Session M25 Suspensions III 328

8:00AM M25.00001 Unifying granular and suspension rheology , FRANCOIS BOYER, ELISABETH GUAZZELLI, OLIVIER POULIQUEN, IUSTI, Aix-Marseille Unversité / CNRS UMR 6595 — Using an original pressure-imposed shear cell, we study the rheology of very dense suspensions. We show that they dexhibit a visco-plastic behavior similarly to granular media successfully described by a frictional rheology and fully characterized by the evolution of the friction coefficient μ and the volume fraction ϕ with a dimensionless viscous number I_v. Dense suspension and granular media are thus unified under a common framework. These results are shown to be compatible with classical empirical models of suspension rheology and provide a clear determination of constitutive laws close to the jamming transition.

8:13AM M25.00002 Suspensions in a tilted trough: second normal stress difference, ELISABETH GUAZZELLI, ETIENNE COUTURIER, FRANCOIS BOYER, OLIVIER POULIQUEN, IUSTI CNRS Aix Marseille University — We measure the second normal-stress difference in suspensions of non-Brownian neutrally-buoyant rigid spheres dispersed in a Newtonian fluid. The second normal-stress-difference is found to be negative and linear in shear stress. The ratio of second normal-stress difference to shear stress increases with increasing volume fraction. A clear behavioural change exhibiting a strong (approximately linear) growth in the magnitude of this ratio with volume fraction is seen above a volume fraction of 0.22. By comparing with previous data obtained for the same batch of spheres in a rotating rod geometry, the ratio of first normal-stress difference to shear stress is estimated and its magnitude is found to be very small.

8:26AM M25.00003 Roles of Particle-Wall and Particle-Particle Interactions in Highly Confined Suspensions of Spherical Particles being Sheared at Low Reynolds Numbers, ASHOK SANGANI[1], National Science Foundation, ANDREAS ACRIVOS, City College of the City University of New York, PHILIPPE PEYLA, Laboratooire de Physique Interdisciplinaire — The roles of particle-wall and particle-particle interactions are examined for suspensions of spherical particles in a viscous fluid being confined and sheared at low Reynolds numbers by two parallel walls moving with equal but opposite velocities. It is shown that the channel-width scale interactions between the spheres tend to decrease the overall viscous dissipation in highly confined suspensions. In other words, the increase in the viscous dissipation caused by the particle-wall interactions is partially compensated by the particle-particle interactions. As a consequence, the total dissipation as a function of particle volume fraction in random suspensions may go through a maximum for a fixed ratio of sphere radius to spacing between the walls.

[1] On leave from Syracuse University

8:39AM M25.00004 Direct experimental investigation of suspension microstructure, BU XU, THARANGA PERERA, JAMES GILCHRIST, Lehigh University — Microstructure is key to understanding rheological behaviors of a flowing particulate suspension. Previous computational results reveal an anisotropic structure forming under high Péclet number conditions. An experimental technique based on confocal microscopy is developed to study microstructure of a colloidal suspension in a microchannel. Results of simulation-like quality are produced and the effect of *Pe* and interparticle force is investigated. Microstructure is shown to be consequential from competitions between hydrodynamic, thermal and electrostatic forces. The stress of suspension, calculated based on the microstructural information, is compared with previous computations.

8:52AM M25.00005 Sedimentation and suspension of a cylinder confined in a Hele Shaw cell, HAROLD AURADOU, Lab. FAST Orsay France, VERONICA D'ANGELO, Grupo de Medios Porosos, Buenos Aires, Argentine, JEAN-PIERRE HULIN, Lab. FAST Orsay France, FRANCO-ARGENTINIAN INTERNATIONAL ASSOCIATED LABORATORY IN THE PHYSICS AND MECHANICS OF FLUIDS COLLABORATION — We present experiences of settling or suspension of a rigid horizontal cylinder moving between two vertical parallel plates. In this study, the ratio between the cylinder diameter and the cell aperture ranges between 0.4 and 0.9 and the Reynolds number is always below the value for vortex shedding at the rear of a fixed cylinder. For ratio below 0.5, the cylinder stays horizontal and translates vertically at a constant velocity. For ratio between 0.5 and 0.7, the cylinder is observed to oscillate in the gap of the cell. This induces a variation of the drag which in turn results in a vertical velocity which oscillates around a mean value. For confinement above 0.7, the cylinder flutters in the Hele Shaw plane. The periodic lateral motion of the cylinder is studied as function of the flow conditions. The dynamic of the system is studied as function of the Reynolds number, confinement and cylinder of various density are also considered. Our observations are compared to a recent study concerning confinement induced vibration for a tethered cylinder.

9:05AM M25.00006 Are falling plumes of particles unstable in Stokes flow?, ANDREW CROSBY, JOHN LISTER, University of Cambridge — A falling plume of heavy particles in viscous flow is observed experimentally to undergo a varicose instability (Pignatel et al., 2009), even though a continuum plume in Stokes flow is known to be stable. We explore this problem through numerical simulations of non-Brownian particle plumes in Stokes flow, varying the average particle number density. An initially cylindrical, axially periodic, particle plume does develop varicose perturbations to its boundary. We demonstrate that this apparent instability is caused by fluctuations in mean particle density, which provide the mechanism for radial perturbations to grow, and determine the growth rate. A competing non-linear wave-breaking mechanism and hydrodynamic diffusion cause the perturbations to saturate and destroy individual bulges. Density fluctuations, wave-breaking and diffusion combine to give a statistically quasi-steady state of bulges on the plume, with much slower diffusive growth of the mean radius. This provides an example of how a background shear flow can greatly reduce hydrodynamic diffusion due to the reversibility of the Stokes equations.

9:18AM M25.00007 Non-monotonic settling of a sphere in a cornstarch suspension, DEVARAJ VAN DER MEER, STEFAN VON KANN, JACCO SNOEIJER, DETLEF LOHSE, University of Twente, NL — Cornstarch suspensions exhibit remarkable behavior. Here, we present two surprising observations for a sphere settling in such a suspension: In the bulk of the liquid the velocity of the sphere oscillates around a terminal value, without damping. Near the bottom the sphere comes to a full stop, but then accelerates again towards a second stop. This stop-go cycle is repeated several times before the object reaches the bottom. We show that common shear thickening or linear viscoelastic models cannot account for the observed phenomena, and propose a minimal jamming model to describe the behavior at the bottom.

9:31AM M25.00008 The instability of sedimenting spheres in a second-order fluid, DAVID SAINTILLAN, RAMANATHAN VISHNAMPET, MechSE, University of Illinois at Urbana-Champaign — The slow sedimentation of a dilute suspension of spherical particles in a second-order fluid is investigated using theory and numerical simulations. A linear stability analysis is performed in the limit of small Deborah number, and shows that such a suspension is unstable to density fluctuations as a result of the nonlinear coupling of the settling motion of the particles under gravity with the local flow field driven by a perturbation in density. Based on this linear theory, an initially homogeneous suspension is expected to develop inhomogeneities, a prediction supported by recent experiments on sedimentation in polymeric liquids. We further confirm this prediction using weakly nonlinear large-scale numerical simulations, which indeed demonstrate the formation of large clusters in the suspensions, resulting in a strong enhancement of the mean settling speed and velocity fluctuations.

9:44AM M25.00009 Accelerated drop detachment in granular suspensions, THIBAULT BERTRAND, CLAIRE BONNOIT, ERIC CLEMENT, ANKE LINDNER, PMMH-ESPCI, PMMH-ESPCI TEAM — We experimentally study the detachment of drops of granular suspensions using a density matched model suspension with varying volume fraction ($\phi = 15\%$ to 55%) and grain diameter ($d = 20\mu m$ to $140\mu m$). We show that at the beginning of the detachment process, the suspensions behave as an effective fluid. The detachment dynamics in this regime can be entirely described by the shear viscosity of the suspension. At later stages of the detachment the dynamics become independent of the volume fraction and are found to be identical to the dynamics of the interstitial fluid. Surprisingly, visual observation reveals that at this stage particles are still present in the neck. We suspect rearrangements of particles to locally free the neck of grains, causing the observed dynamics. Close to the final pinch off, the detachment of the suspensions is further accelerated, compared to the dynamics of pure interstitial fluid. This acceleration might be due to the fact that the neck diameter gets of the order of magnitude of the size of the grains and a continuous thinning of the liquid thread is not possible any more. The crossover between the different detachment regimes is function of the grain size and the initial volume fraction. We characterize the overall acceleration as a function of the grain size and volume fraction.

9:57AM M25.00010 Correlation between particle diffusion and effective heat transfer in a sheared suspension[1] , XIAOLONG YIN, Petroleum Engineering, Colorado School of Mines, BLOEN METZGER, IUSTI CNRS, Polytech Marseille — We show that enhanced solute or heat transport observed in sheared suspensions of neutrally buoyant particles with low Reynolds numbers are well correlated to the particle diffusion. Experiments were conducted in a Couette cell where the effective heat transfer coefficients were extracted from the decay of a heat pulse applied to a bounding wall. The particle diffusion coefficients were measured by particle tracking enabled by laser-induced fluorescence and refractive index matching. A numerical method based on a combination of the lattice Boltzmann method and Brownian tracers was developed to simulate the convective transport process. Effective heat transfer coefficients were obtained from matching the tracer profile with a transient analytical solution of the heat transfer equation. Heat transfer barriers were identified near the walls and the particle surfaces due to lack of convective motion in the direction normal to the surfaces. Both experimental and numerical data show that the effective heat transfer coefficient increases substantially with increasing shear rate, and the dependence on the volume fraction shows a peak at about 35% volume fraction that coincides with the peak in the particle diffusion coefficients.

[1]Visits and experiments were supported by ANR JCJC SIMI 9

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M26 Nanofluids III 329

8:00AM M26.00001 Laminar Channel Flow Over Superhydrophobic Porous Surfaces , OZGUR OZSUN, VICTOR YAKHOT, KAMIL L. EKINCI, Boston University — We have fabricated mesh-like porous superhydrophobic surfaces with solid area fraction ϕ_s, which can maintain intimate contact with both air and water reservoirs on either side. Typical structure has linear dimensions of $1mm \times 1mm \times 200nm$ and pore area of $10\mu m \times 10\mu m$. Recent experiments on such surfaces have shown anomalous hydrodynamic response in oscillating flows [1]. This effect is attributed to a Knudsen layer of gas at the solid-liquid interface. In this study, we investigate laminar channel flow over porous superhydrophobic surfaces. The surfaces are enclosed with PDMS microchannels, where pressure driven flow of DI water is generated. Pressure drop across the microchannels is measured as a function of flow rate. Slip lengths are inferred from the Poiseuille relation as a function of ϕ_s and compared to that of similar standard superhydrophobic surfaces, which lack intimate contact with air reservoir.

[1] http://arxiv.org/abs/1107.5873

8:13AM M26.00002 Micron-scale measurement of dynamic contact angles in the vicinity of moving contact lines , BIAN QIAN, KENNETH BREUER, Brown University — Directly probing the liquid-vapor interface shape near moving contact lines is critical to understanding the microscopic physics of dynamic contact line behavior. Using a novel high resolution optical technique, we report on the meniscus shape in the immediate vicinity of moving contact lines. The liquid is fluorescently-dyed and illuminated with a highly focused laser beam. The strongest illumination intensity occurs at the laser focus and this bright and tiny spot serves as a probe which changes in intensity and shape near the liquid-vapor interface. This experimental setup enables us to resolve the meniscus shape with submicron resolution within 10 microns of the contact line. Using this technique, we report on the dependence of the interface shape on the contact line speed and compare it with theoretical predictions.

8:26AM M26.00003 Uncertainty Quantification in MD Simulations: Forward Propagation and Parameter Inference[1] , FRANCESCO RIZZI, OMAR KNIO, Johns Hopkins University, HABIB NAJM, BERT DEBUSSCHERE, KHACHIK SARGSYAN, MAHER SALLOUM, HELGI ADALSTEINSSON, Sandia National Laboratories — This work focuses on quantifying uncertainty in molecular dynamics (MD) simulations accounting for both intrinsic noise and parametric uncertainty. We consider isothermal, isobaric MD simulations of TIP4P water at ambient conditions. Due to the thermal noise in the system, this yields non-deterministic, noisy predictions for the water observables. Selected macroscale observables are expressed in terms of polynomial chaos (PC) expansions using both a non-intrusive spectral projection (NISP) and Bayesian inference approach. We show that the effect of the thermal noise can be controlled, and that the two methods yield similar results. We illustrate the possibility of determining or refining a set of force-field parameters for water using a reformulated Bayesian approach based on PC expansions. We present a synthetic problem where presumed "true" values of the TIP4P model parameters are used to generate a collection of noisy data of selected water observables. Exploiting the PC representation, we show how the "true" force-field parameters can be accurately and efficiently recovered using low-order surrogate models.

[1]Work supported by the U.S. Department of Energy, Office of Advanced Scientific Computing Research.

8:39AM M26.00004 Molecular Dynamic Simulation of Flow in a Channel with Oscillating Wall , JOSEPH THALAKKOTTOR, KAMRAN MOHSENI, University of Florida — Molecular dynamic simulations are employed in order to investigate the effect of fluid inertia on slip at a solid surface. The accuracy of the numerical technique is verified by reproducing the steady state slip boundary condition for Couette flow at a solid surface reported by Thompson and Troian, Nature 1997. Numerical experiments are also conducted on Couette flow with oscillatory wall velocity in order to investigate the effect of fluid inertial on the slip boundary condition. Preliminary results indicate that the accelerating/decelerating wall velocities could have some effect of the slip value at the wall.

8:52AM M26.00005 Wall Boundary Conditions in 3D Smooth Dissipative Particle Dynamics Simulations[1] , JUN YANG, RAFFAELE POTAMI, NIKOLAOS GATSONIS — The mesh-free thermodynamically consistent smooth dissipative particle dynamics (SDPD) method is implemented in 3D using a block structure to minimize computational time and the Verlet neighbor-list algorithm for efficient search of particles. A new method for solid wall boundary conditions is presented that enforces non-penetration and minimizes the density fluctuations. The geometry of the wall is described using wall pseudo-particles. This geometry is used to generate a set of ghost fluid particles obtained by mirroring the neighbor particles of a given fluid particle in the proximity of a boundary. The ghost particles are used for minimizing the density fluctuations near walls, for generating the repulsive force that ensures the impenetrability condition, and for generating a dissipative force that ensure the no slip condition. Numerical results for isothermal flows show only minimal density fluctuations in the proximity of the solid wall and an overall good agreement between the velocity profiles from numerical simulations and analytical results.

[1]Sponsored by the Computational Mathematics Program of the Air Force Office of Scientific Research under grant/contract number FA9550-06-1-0236.

9:05AM M26.00006 Behavior of the Kapitza Resistance at Liquid/Solid Interfaces: Insights from Molecular Dynamics and Continuum Studies , ANOOSHEH NIAVARANI, SANDRA TROIAN, California Institute of Technology, 1200 E. California Blvd. MC 128-95, Pasadena, CA — While Kapitza (1941) first proposed the concept of thermal boundary resistance in superfluid helium, Khalatnikov (1952) soon realized that this phenomenon was caused by acoustic phonon mismatch across any interface separating dissimilar materials. Acoustic phonon mismatch has also been invoked as the source of instability in viscous nanofilms exposed to a large thermal gradient [1]. Liquid films approaching nanoscale dimensions are especially prone to thermal boundary resistance effects which can substantially alter the internal temperature profile. For this purpose, non-equilibrium molecular dynamics (NEMD) simulations provide an especially convenient tool for investigation [2-4]. Using NEMD and continuum studies, we explore the origins of the discontinuity in temperature at liquid/solid interfaces and specify regimes leading to deviations from Fourier's law. These findings suggest that although thermal convection effects are often negligible in nanofluidic systems, the Kaptiza resistance can nonetheless strongly influence hydrodynamic flow at small scales. [1] E. Schaffer et al., Macromolecules 36, 1645 (2003) [2] L. Xue et al., Int. J. Heat Mass Transfer 47, 4277 (2004) [3] B. Kim, A. Beskok and T. Cagin, J. Chem. Phys. 129, 174701 (2008) [4] S. Murad and I. Puri, Appl. Phys. Lett. 92, 133105 (2008)

9:18AM M26.00007 Submicron flows of polymer solutions , AMANDINE CUENCA, HUGUES BODIGUEL, LOF UMR5258, University of Bordeaux 1, CNRS, Rhodia 178, avenue du Docteur Schweitzer, 33608 Pessac - France — We study flow properties of high molecular weight polymer solutions below the micrometer scale. Fluorescence photobleaching is used as a non-invasive technique to evaluate the velocity of pressure-driven flows in channels from 200 to 4000 nm height. We observe a striking reduction of the effective viscosity. The latter is reduced up to 50 times at 400 nm. This effect increases with molecular weight and concentration. Using a Rabinowitsch-like approach, we correlate the data at different thicknesses to obtain both the slippage at the wall and the rheological flow curve at sub-microscale. Those properties are also evaluated in bulk using respectively PIV (Particle Image Velocimetry) and rheometer. Comparing the measurements in bulk and in confined geometries, we conclude that the viscosity reduction can not solely be explained by slippage. We can think of two origins of the confinement effect, either it modifies the polymer solution on itself, in terms of concentration (size-dependent filtration of coils) or it induces a change in the rheological behaviour.

9:31AM M26.00008 Ultra-fast Thinning of Nanoscale Films by Convection , MARKUS ABEL, Potsdam University, MICHAEL WINKLER, Institute for Physics and Astronomy, Potsdam University, Germany, RUMEN KRASTEV, NMI Natural and Medical Science Institute, Reutlingen, Germany — We present experimental results on flows on very thin, nanometer–scale membranes. More specific we observe an enormous speed-up of the thinning of a film with surfactants towards its equilibrium of a few nanometers when we drive it thermally by cooling locally on a spot in the upper third of the film. Interesting and beautiful to watch by itself, the thinning of films is one of the most important topics for applications and research: Interfaces in general are important for all two-phase flows, as for colloids, aerosols, droplets, or for the production of ultra-thin materials, etc.. The involved processes are macroscopic in 2 dimensions and at nanometers in the third one. Consequently, the system is just at the edge of the hydrodynamic description; furthermore, the chemical and electrical forces of the surfactant and bulk material become important. We have combined convection with thin films in order to observe the effects of thermal driving on thinning and vice versa. As a result, mixing was observed which does not only accelerate the thinning but changes the thinning law qualitatively from linear to expoential.

9:44AM M26.00009 Near wall velocity measurement of nanofluids using evanescent wave based PIV technique[1] , ANOOP KANJIRAKAT, REZA SADR, Texas A&M University at Qatar — Recently nanofluids, dilute suspension of nano particles in a based fluid, have emerged as a possible candidate as a coolant in variety of applications based on some reports of enhanced conductivity for these fluids. However, there are controversies in the reported properties of nanofluids and their applicability's, specially, since there is no fundamental understanding explaining these enhancements. A better understanding of these fluids and how they interact with the solid boundary is achieved by detailed near wall fluid flow study at nano scale. In this work nanofluids of different concentrations are prepared by dispersing silicon dioxide particles (10-20nm) in water as the base fluid. Pressure driven nanofluids flow inside a micro channel is studied by adding fluorescent polystyrene particles of 100nm to the flow. Nano Particle Image Velocimetry (nPIV) is used to measure near-wall velocity fields with an out of plane resolution of less than 250nm. The measured near wall velocity field is then compared with that of the basefluid at the same condition. Initial observations have shown that nPIV techniques can be successfully extended for nanofluids for better understanding the flow characteristics in the near wall region.

[1]This work was supported by QNRF.

9:57AM M26.00010 Surface Nanobubble Stability , JAMES SEDDON, University of Twente, HAROLD ZANDVLIET, DETLEF LOHSE — We provide a model for the remarkable stability of surface nanobubbles to bulk dissolution. The key to the solution is that the gas in a nanobubble is of Knudsen type. This leads to the generation of a bulk liquid flow which effectively forces the diffusive gas to remain local. Our model predicts the presence of a vertical water jet immediately above a nanobubble, with an estimated speed of $\sim 3.3m/s$, in good agreement with our experimental atomic force microscopy measurement of $\sim 2.7m/s$. In addition, our model also predicts an upper bound for the size of nanobubbles, which is consistent with the available experimental data.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M27 Microfluids: General VI Ballroom I

8:00AM M27.00001 Streaming potential revisited , EHUD YARIV, ORY SCHNITZER, ITZCHAK FRANKEL, Technion — Streaming-potential phenomena refer to the generation of bulk electric fields by imposed relative motion between a charged solid and the Debye layer adjacent to it. Realistic scenarios are adequately described by the thin-Debye-layer limit $\delta \to 0$ (δ denoting the dimensionless Debye thickness), which has been addressed by Cox (1997). Cox's analysis has established that the perturbation to the flow, neglected in the earlier investigations, gives rise to an $O(\delta^4)$ force that dominates that contributed by Maxwell stresses. Cox's theory is founded upon the assumption of $O(1)$ Hartmann and Péclet numbers. We demonstrate that the product of these numbers is actually $O(\delta^{-2})$ and accordingly revisit the generic problem of streaming-potential. Electric-current matching between the Debye layer and the bulk provides an inhomogeneous Neumann condition governing the electric field in the latter. This field, in turn, results in a velocity perturbation animated by a Smoluchowski-type slip condition. Owing to dominant convection, the present analysis yields an asymptotic structure considerably simpler than that of Cox (1997): the electro-viscous effect now already appears at $O(\delta^2)$ and is contributed by both Maxwell and viscous stresses. The present paradigm is illustrated for the prototypic problem of a sphere sedimenting in an unbounded fluid, with the resulting drag correction differing from that calculated by Cox (1997).

8:13AM M27.00002 Electrokinetic lift , ORY SCHNITZER, EHUD YARIV, ITZCHAK FRANKEL, Technion — Electrolyte flow relative to a charged surface induces a bulk electric field (the "streaming potential" phenomena). This field, and the flow perturbation it animates, generate both electrical and hydrodynamic "electro-viscous" forces whose magnitude has been a matter of ongoing controversy. Recently we have revisited this problem, predicting $O(\delta^2)$ scaling (as opposed to earlier prediction of δ^4 and δ^6), $\delta \ll 1$ being the dimensionless Debye width. These electro-viscous forces can explain the anomalous repulsion of polystyrene microspheres from an adjacent wall in the presence of an imposed shear flow, observed by Prieve and co-workers. Owing to the symmetry properties of the linear Stokes equations, such repulsion is inadmissible in the absence of inertial effects. This particle–wall interaction is analyzed using our revised scheme. The undisturbed flow consists of three components: the 'driving' shear mechanism and the 'induced' particle translation and rotation. We consider a small dimensionless particle–wall gap ϵ. At leading-order, both the lift and additional drag are contributed by the inner gap region. The lift force is $O(\delta^2\epsilon^{-3})$ while the additional drag is $O(\delta^2\epsilon^{-2})$. The streaming-potential mechanism underlying these forces arises from the 'induced' rather than the driving component.

8:26AM M27.00003 From diffusive motion to sub-diffusive motion with local aggregation: the effect of surface contamination in dipolophoresis , JAE SUNG PARK, DAVID SAINTILLAN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — We investigate the effects of surface contamination in suspensions of ideally polarizable spheres in an applied electric field using large-scale particle simulations. In the case of clean particles, suspensions are known to undergo dipolophoresis (DIP), or combination of dielectrophoresis (DEP) and induced- charge electrophoresis (ICEP), in which ICEP dominates the dynamics over DEP. Surface contamination is modeled as a thin dielectric layer coated uniformly around the particle surface, which can be captured by a means of a correction factor multiplying the induced zeta potential or slip velocity driving ICEP. Because the correction factor is always less than unity, ICEP is gradually suppressed as surface contamination becomes significant, resulting in a transition in the suspensions from diffusive motions to sub-diffusion and local aggregation. Large- scale simulation results are presented on the suspension microstructure, as well as on particle velocities, which are strongly reduced for contaminated particles. We demonstrate a clear transition from diffusion to sub-diffusion by calculating particle mean-square displacements, and we explain it as a consequence of the non-integrable local waiting time distributions that arise in the sub-diffusive regime due to the trapping of the particles inside chains and clusters.

8:39AM M27.00004 Directly resolving particles in an electric field: local charge, force, torque, and applications[1] , QIANLONG LIU, DOE Energy Frontier Research Center, University of South Carolina — Prosperetti's seminal Physalis method for fluid flows with suspended particles is extended to electric fields to directly resolve finite-sized particles and to investigate accurately the mutual fluid-particle, particle-particle, and particle-boundary interactions. The method can be used for uncharged/charged dielectrics, uncharged/charged conductors, conductors with specified voltage, and general weak and strong discontinuous interface conditions. These interface conditions can be in terms of field variable, its gradients, and surface integration which has not been addesed by other numerical methods. In addition, for the first time, we rigorously derive the force and torque on the finite-sized particles resulting from the interactions between harmonics. The method, for the first time, directly resolves the particles with accurate local charge distribution, force, and torque on the particles, making many applications in engineering, mechanics, physics, chemistry, and biology possible, such as heterogeneous materials, microfluidics, electrophotography, electric double layer capacitors, and microstructures of nanodispersions. The efficiency of the method is demonstrated with up to one hundred thousand 3D particles, which suggests that the method can be used for many important engineering applications of broad interest.

[1]This research is supported by the Department of Energy under funding for an EFRC (the HeteroFoaM Center), grant no. DE-SC0001061.

8:52AM M27.00005 Multiscale study of nanoparticle-wall interactions in electroosmotic flow[1] , A.T. CONLISK, HARVEY ZAMBRANO, ZHIZI PENG, The Ohio State University — In electroosmotic transport (EOT), particle mobility results not only from the dragging exerted by the electrolyte, but also from the force exerted by the External Electric Field (EEF), and from the interactions with the walls and with the solvent. The objective of this work is to develop a unified theory of the motion of colloidal particles near walls and compare with the experiments of Kazoe and Yoda for EOT. In the present study a novel continuum approach is developed to study the particle interactions with polystyrene beads. Moreover, we conduct Non-equilibrium Molecular Dynamics Simulations (NEMDS) of a nanoparticle as it moves near a solid-liquid interface subjected to an EEF. We investigate the response of the particle to changes in the surface electrostatics and the electrolyte concentration. Therefore, we perform NEMDS of a silica particle immersed in an electrolyte. The electrolyte solution is mounted on a silica substrate and the particle is constrained to move parallel to the surface so that we can extract the forces acting between the particle and the wall. We vary the electrolyte concentration, the particle size and the surface electrostatics.

[1]Supported by the Army Research Office, the National Science Foundation NSEC Center for the Affordable Nanoengineering of Polymeric Biomedical Devices

9:05AM M27.00006 Role of Background Electrolyte in Electrokinetic Locomotion by Reaction-Induced Charge Auto-Electrophoresis , JEFFREY L. MORAN, JONATHAN D. POSNER, University of Washington — Bimetallic particles propel themselves through aqueous solutions by harvesting chemical energy from hydrogen peroxide fuel and converting it to fluid motion through reaction-induced charge auto-electrophoresis. We present a scaling analysis and computational simulations that describe the physics underlying the locomotion of these particles. The model shows that the motion results from electrical body forces in the surrounding fluid, which are generated by a coupling of an asymmetric dipolar charge density distribution and the electric field it generates. The simulations and scaling analysis make the predictions, in agreement with experiments, that the speed of the autonomous motion depends linearly on fuel concentration and particle surface charge and inversely on solution conductivity.

9:18AM M27.00007 Electrokinetically induced flocculation of Enteroaggregative *Escherichia coli* , ALOKE KUMAR, NINELL MORTENSEN, Biosciences Division, Oak Ridge National Laboratory, Oak Ridge, TN 37831, MANSUETA HARRIS, Division of Natural Sciences and Mathematics, Morris College, Sumter, SC 29150, PARTHA MUKHERJEE, Computer Science and Mathematics Division, Oak Ridge National Laboratory, Oak Ridge, TN 37831, SCOTT RETTERER, MITCHEL DOKTYCZ, Biosciences Division, Oak Ridge National Laboratory, Oak Ridge, TN 37831 — Enteroaggregative *Escherichia coli* (EAEC) is a diarrheal microbe, whose aggregative dynamics is involved in its pathogenic behavior. We investigated EAEC's electrokinetic response in miniaturized and microfluidic devices. We found a novel response of the microbe under low magnitude, uniform and oscillating electric fields. In this electrokinetically induced response, microbial adhesion to a glass substrate decreases significantly, leading to a loss of EAEC's biofilm forming abilities. Some earlier studies had indicated that that microbial adhesion and detachment at surfaces can be prompted only by charge-transfer processes at the electrode and not applied electrical potentials - such an inference is not corroborated by our work. Instead, we found that electric fields promote the formation of large mesoscopic microbial aggregations (flocs) in the solution. The presence of frequency dependent relaxation phenomena is explored and the observed results are extended to other microbes.

9:31AM M27.00008 Magnetoviscosity and thread-like agglomerations in ferrofluids[1] , PHILIP YECKO, A. CALI, Montclair State University, Montclair NJ, W.-K. LEE, Advanced Photon Source, Argonne National Laboratory, S. NUNEZ, J. PRESCOD, R. SMITH, A.D. TRUBATCH, M. VIEIRA, Montclair State University, Montclair NJ — We report on experiments and simulations performed on small non-magnetic glass balls falling under gravity through a magnetized ferrofluid. The applied magnetic field is oriented horizontally, normal to the fall, and is uniform but its magnitude can be adjusted over a wide range. Using the Advanced Photon Source x-ray beamline at Argonne, we were able to achieve sufficient spatial and temporal resolution to track the dynamics of these 500μm diameter spheres simultaneously with an array of magnetic particle *macro-chains* – thread-like agglomerations each several mm long and $2-10\mu$m thick. The enhanced drag induced by the macro-chains is enormous: up to four times larger than for unmagnetized fluid, a value greater than is predicted by the prevailing magneto-viscosity model. We provide direct visualization of a possible mechanism by which macro-chains impede the transverse motion of spheres. Numerical simulations can reproduce the observed drag, without modeling it physically, by implementing a simple magnetization dependent anisotropic viscosity.

[1]Supported by NSF grant no. MPS-1016383.

9:44AM M27.00009 Non-Clogging Resistive Pulse Sensing with Electrohydrodynamic Cone-Jet Bridges , CHUAN-HUA CHEN, YUEJUN ZHAO, DAVID BOBER, Duke University — A new paradigm of resistive pulse sensing (Coulter counting) is developed using a liquid bridge in lieu of a solid pore as the sensing aperture, where the flexible liquid aperture circumvents the clogging issue of conventional Coulter counters. The electrohydrodynamic bridge is formed between two opposing Taylor cones and stabilized by radial polarization stresses. Passage of a colloidal particle through the upstream conical apex triggers a current oscillation at the resonant frequency of the cone-jet bridge. The relative current change is indicative of the particle- to-jet diameter ratio.

9:57AM M27.00010 ABSTRACT HAS BEEN MOVED TO M29.00009 —

Tuesday, November 22, 2011 8:00AM - 9:57AM –

Session M28 Undulatory Locomotion Ballroom II

8:00AM M28.00001 Lightweight robot locomotion on granular media , TINGNAN ZHANG, FEIFEI QIAN, JEFFREY SHEN, CHEN LI, Georgia Institute of Technology, AARON HOOVER, Olin College, PAUL BIRKMEYER, RONALD FEARING, University of California at Berkeley, DANIEL GOLDMAN, Georgia Institute of Technology — We present an experimental and computer simulation study of a small, light-weight, biologically inspired robot running on a model granular medium (GM), 3 mm diameter glass particles. The six-legged RoACH robot (10 cm long, 25 grams) utilizes an alternating tripod gait to run at speeds up to 25 cm/sec. Forward speed increases with increasing limb frequency $0 < f < 12$ Hz. An experimentally validated discrete element method (DEM) simulation of the device captures the observed mechanics. Observation from high speed video and simulation reveals that at low f, there is little slip of the limb through the GM, and forward speed is set by step length. At higher f, limbs slip continuously through the GM and fluidize the surrounding material. In this regime, speed is dominated by fluid-like thrust forces.

8:13AM M28.00002 Motility analysis of the nematode C. elegans on wet soft media , JOSUE SZNITMAN, Technion - Israel Institute of Technology, XIAONING SHEN, PAULO ARRATIA, University of Pennsylvania — Undulatory locomotion is widely utilized by limbless organisms such as snakes, eels and worms. When moving on top of wet soft gels (e.g. agar), undulating organisms such as the nematode *Caenorhabditis elegans* display a motility gait that is characterized by crawling. Until present however, a detailed understanding of how *C. elegans*' crawling gait generates propulsion over soft gels is lacking. Namely, how much crawling force does *C. elegans* generate? Here, we propose a simple model based on lubrication theory to examine the biomechanics of crawling motion. In analogy to the well-known resistive-force theory (RFT) for low Reynolds number swimming, our model provides a mechanism for the linear relation between the sliding speeds and the drag forces, and sheds light on the role of grooves created by nematodes on agar. We further examine the kinematics of locomotion experimentally and compare muscle activity patterns between crawling and swimming gaits, emphasizing the inherent differences in nematode adaptability to different environments.

8:26AM M28.00003 Undulating Underperformance: Swimming in Elastic Media[1] , XIAONING SHEN, PAULO ARRATIA, University of Pennsylvania — In this talk, we investigate the effects of fluid elasticity on the swimming behavior of the nematode *Caenorhabditis elegans* by tracking the nematode's motion and measuring the corresponding velocity fields. We find that fluid elasticity hinders self-propulsion and fluid transportation. Compared to Newtonian solutions, fluid elasticity leads to 35% slower propulsion speed. Furthermore, self-propulsion and fluid transportation are weakened as elastic stresses grow in magnitude in the fluid. This decrease in self-propulsion in viscoelastic fluids is related to the stretching of flexible molecules near hyperbolic points in the flow.

[1]This work was supported by NSF-CAREER (CBET)-0954084.

8:39AM M28.00004 Locomotion of C elegans in structured environments , TRUSHANT MAJMUDAR, ERIC KEAVENY, MICHAEL SHELLEY, JUN ZHANG, Courant Institute, New York University — We have established a combined experimental and numerical platform to study the swimming dynamics of an undulating worm in structured environments (fluid-filled micro-pillar arrays). We have shown that the worm (C. elegans) swims with different velocity and frequency depending on the lattice spacing and our purely mechanistic simulations (elastically linked bead-chain) reproduce the experimental results qualitatively and quantitatively, including "life-like" trajectories the worm exhibits. We build upon this platform to investigate more complex environments, such as linear and radial lattices, with gradients in spacing. In addition, we study C. elegans mutants to investigate the role of length of the worm, frequency of undulations, and mechano-sensation on the resultant dynamics. We also examine the worm moving through a lattice with random distribution of obstacles - a model soil-like environment. Our combined experimental and simulations approach allows us to gain insights into the dynamics of locomotion of undulating microorganisms in realistic complex environments.

8:52AM M28.00005 A Treadmill to Localize, Exercise, and Measure the Propulsive Power of Nematodes , JINZHOU YUAN, HAN-SHENG CHUAN, MICHAEL GNATT, DAVID RAIZEN, HAIM BAU, University of Pennsylvania — The nematodes *C. elegans* is often used as model biological system to study the genetic basis of behavior, disease-progression, and aging, as well as to develop new therapies and screen drugs. On occasion, it is desirable to quantify the nematode's muscle power. Here, we present a kind of nematode treadmill. The device consists of a tapered conduit filled with aqueous solution. The conduit is subjected to a DC electric field and to pressure-driven flow directed from the narrow end. The nematode is inserted at the conduit's wide end. Directed by the electric field (through electrotaxis), the nematode swims deliberately upstream toward the negative pole. As the conduit narrows, the average fluid velocity and the drag force on the nematode increase. Eventually, the nematode arrives at an equilibrium position, at which its propulsive power balances the viscous drag force. The nematode's propulsive power is estimated with direct numerical simulations of the flow field around the nematode. The calculations utilize the experimentally imaged gait as a boundary condition. The device is useful to retain the nematode at a nearly fixed position for prolonged observations under a microscope, to keep the nematode exercising, and to estimate the nematode's power based on the conduit's width at the equilibrium position.

9:05AM M28.00006 Propulsion of *C. elegans* crawling on a wet surface[1] , A. BILBAO, Texas Tech University, A. ALAVALAPADU, Texas Tech HSC, Z.S. KHAN, Texas Tech University, D.E. SALOMON, Texas Tech Univeristy, S.A. VANAPALLI, Texas Tech University, K. RUMBAUGH, Texas Tech HSC, J. BLAWZDZIEWICZ, Texas Tech University — Nematodes, such as soil-dwelling worms C. elegans, propel themselves by producing undulatory body motion. An important requirement for effective propulsion is to have large transverse and small longitudinal friction forces acting on a crawling worm. Recently, Sauvage et al. have shown that soft-lubrication forces between the worm body and a moist supporting substrate can produce, at most, the transverse friction coefficient twice as large as the longitudinal friction coefficient (and this ratio is too small for efficient propulsion). Here we show that hydrodynamic resistance of the fluid in liquid film adjacent to the worm body can generate significantly larger transverse friction, which moreover, is wavelength dependent. By modeling the worm as a long chain of spheres in Hele–Shaw flow, we have determined the optimal wavelength and amplitude of the undulatory motion that optimizes propulsion efficiency for a given rate of energy dissipation. The optimal worm shape qualitatively agrees with our experimental observations of C. elegans crawling in moist environments.

[1]This work was supported by NSF Grant No. CBET-1059745.

9:18AM M28.00007 Three-link Swimming in Sand , R.L. HATTON, Carnegie Mellon University, YANG DING, ANDREW MASSE, Georgia Institute of Technology, HOWIE CHOSET, CMU, DANIEL GOLDMAN, Georgia Tech — Many animals move within in granular media such as desert sand. Recent biological experiments have revealed that the sandfish lizard uses an undulatory gait to swim within sand. Models reveal that swimming occurs in a frictional fluid in which inertial effects are small and kinematics dominate. To understand the fundamental mechanics of swimming in granular media (GM), we examine a model system that has been well-studied in Newtonian fluids: the three-link swimmer. We create a physical model driven by two servo-motors, and a discrete element simulation of the swimmer. To predict optimal gaits we use a recent geometric mechanics theory combined with empirically determined resistive force laws for GM. We develop a kinematic relationship between the swimmer's shape and position velocities and construct connection vector field and constraint curvature function visualizations of the system dynamics. From these we predict optimal gaits for forward, lateral and rotational motion. Experiment and simulation are in accord with the theoretical predictions; thus geometric tools can be used to study locomotion in GM.

9:31AM M28.00008 The role of body stiffness in wake production for anguilliform swimmers[1] , ERIC TYTELL, Johns Hopkins University, MEGAN LEFTWICH, Los Alamos National Laboratory, CHIA-YU HSU, Feng Chia University, AVES COHEN, University of Maryland, College Park, LISA FAUCI, Tulane University, ALEXANDER SMITS, Princeton University — We compare wake structures shed by the undulatory motion of physical and computational models of an anguilliform swimmer, the lamprey. The physical model is a robotic lamprey-like swimmer with an actively flexing tail, and with passively flexible tails of different stiffnesses. The computational model is a two-dimensional computational fluid dynamic (CFD) model that captures fluid-structure interaction using the immersed boundary framework. The CFD model included both actively flexing and passively flexible tail regions. Both models produced wakes with two or more same-sign vortices shed each time the tail changed direction (a "2P" or higher- order wake). In general, wakes became less coherent as tail flexibility increased. We compare the pressure distribution near the tail tip and the timing of vortex formation in both cases and find good agreement. Differences between self-propelled and tethered cases are detailed. Finally, we examine the effects of material resonance on force production.

[1]Supported by NIH R01 NS054271.

9:44AM M28.00009 Propulsion of an undulating elastic filament on a free surface[1] , SOPHIE RAMANANARIVO, BENJAMIN THIRIA, RAMIRO GODOY-DIANA, PMMH UMR7636 CNRS; ESPCI ParisTech; UPMC; U. Diderot Paris 7 — Nature offers a lot of examples of swimmers that use body undulations to move forward, such as eels or sperm. This type of locomotion involves strong fluid-structure interactions regardless of the regime of Reynolds number considered. Here we study a flexible filament forced to oscillate by imposing a harmonic motion to one of its extremities (using magnetic interactions) and propelling itself at the surface of a water tank. This experiment serves as a canonical model for studying the interactions between an elastic structure undergoing complex deformations and the surrounding fluid. We characterized the nature of the wave travelling the filament (by measuring its amplitude, phase velocity and spatial damping), as well as its propulsive performance in the different regimes encountered.

[1]We acknowledge support from the French National Research Agency through project No. ANR-08-BLAN-0099

Tuesday, November 22, 2011 8:00AM - 10:10AM –

Session M29 Biofluids: Cellular V: Complex Interactions Ballroom III

8:00AM M29.00001 Effect of physical variables on capture of magnetic nanoparticles in simulated blood vessels[1] , MINGHUI ZHANG, Carnegie Mellon University, CHRISTOPHER BRAZEL, University of Alabama — This study investigated how the percent capture of magnetic nanoparticles in a simulated vessel varies with physical variables. Magnetic nanoparticles (MNPs) can used as part of therapeutic or diagnostic materials for cancer patients. By capturing these devices with a magnetic field, the particles can be concentrated in an area of diseased tissue. In this study, flow of nanoparticles in simulated blood vessels was used to determine the affect of applying an external magnetic field. This study used maghemite nanoparticles as the MNPs and either water or Fetal Bovine Serum as the carrier fluid. A UV-Vis collected capture data. The percent capture of MNPs was positively influenced by five physical variables: larger vessel diameters, lower linear flow velocity, higher magnetic field strength, better dispersion, lower MNP concentration, and lower protein content in fluid. Free MNPs were also compared to micelles, with the free particles having more successful magnetic capture. Four factors contributed to these trends: the strength of the magnetic field's influence on the MNPs, the MNPs' interactions with other particles and the fluid, the momentum of the nanoparticles, and magnetic mass to total mass ratio of the flowing particles.

[1]Funded by NSF REU Site #1062611.

8:13AM M29.00002 Viscoelastic Response of Cytoskeleton: Prestress effect[1] , LEN PISMEN, KONSTANTIN MOROSOV, MARCO SALM, Department of Chemical Engineering, Technion - Israel Institute of Technology — The differential elastic modulus of an active actomyosin network is computed as a function of applied stress, taking into account both thermal and motor contributions to filament compliance in the low-frequency domain. It is shown that, due to a dual nature of motors activity, increasing motor concentration may either stiffen the network due to stronger prestress or soften it due to motor agitation, in accordance with experimental data. Prestress anisotropy, which may be induced by redistribution of motors triggered by external force, causes anisotropy of elastic moduli. This helps to explain contradictory phenomena of cell fluidization and resolidification in response to transient stretch. Reshaping of epithelial cells is modeled by allowing for prestress anisotropy due to myosin redistribution.

[1]This work has been supported by the Human Frontier Science Program (Grant RGP0052/2009-C).

8:26AM M29.00003 Numerical study on the initial stage of thrombus growth , SHU TAKAGI, SATOSHI II, The University of Tokyo, SEIJI SHIOZAKI, RIKEN, KAZUYASU SUGIYAMA, YOICHIRO MATSUMOTO, The University of Tokyo — Thrombosis is regarded as one of the most important diseases, which cause the myocardial and cerebral infarctions. It is affected from molecular scale protein-protein interaction to continuum scale in blood flow. Initially, platelets start aggregate at the injured vessel wall, where von Willebrand Factor (vWF) is attached. The Glycoprotein, GPIb-αs on platelet membrane starts showing ligand-receptor interaction with this vWF and platelets start aggregating around this spot. In the present study, the molecular scale interaction between vWF and GPIb-α g is taken into account through the kinetic Monte Carlo simulations. Then, the interacting force between platelets and vascular endothelium obtained from kinetic Monte Carlo simulation is coupled with the continuum scale simulation. The results illustrate that platelets are much easier to aggregate on the wall in the presence of red blood cells and the effect of molecular interaction force are quantitatively discussed on the aggregation of platelets.

8:39AM M29.00004 Diamagnetic Cell Focusing in Ferrofluid Microchannel Flows , JIAN ZENG, LITAO LIANG, TZUEN-RONG TZENG, XIANGCHUN XUAN, Clemson University — Focusing cells into a tight stream is usually a necessary step prior to counting, detecting, and sorting them. It has been achieved by using either sheath flow(s) to pinch the cellular stream or force(s) to manipulate the suspended cells directly. In this talk we present a three-dimensional magnetic cell focusing method in ferrofluid flow through a rectangular microchannel. A pair of facing permanent magnets is embedded into the chip to create a magnetic field null inside the microchannel. Diamagnetic cells experience negative magnetophoresis in a more magnetizable ferrofluid and thus migrate toward the centerline of the bottom channel wall. The effects of flow rate and ferrofluid concentration on the magnetic focusing of yeast cells in both the horizontal and the vertical planes of the microchannel are examined experimentally. The obtained results are compared with the theoretical predictions of an analytical model.

8:52AM M29.00005 Cellular Dewetting: Opening of Macroapertures in Endothelial Cells[1] , DAVID GONZALEZ-RODRIGUEZ, Institut Curie, MADHAVI MADDUGODA, CAROLINE STEFANI, Universite de Nice, SEBASTIEN JANEL, FRANK LAFONT, Institut de Biologie de Lille, DAMIEN CUVELIER, Institut Curie, EMMANUEL LEMICHEZ, Universite de Nice, FRANCOISE BROCHARD-WYART, Institut Curie — Pathogenic bacteria can cross from blood vessels to host tissues by opening transendothelial cell macroapertures. Here we model the opening of macroapertures as a new form of dewetting, driven by the cell's membrane tension. While liquid dewetting is irreversible, we show that cellular dewetting is transient. Our model predicts the minimum radius for hole nucleation, the maximum hole size, and the dynamics of opening, in good agreement with the experiments. The physical model is then coupled with biological experimental data to reveal that a certain curvature-sensing protein controls the line tension at the rim of the hole and opposes its opening.

[1]D.G.-R. is supported by Fondation Pierre-Gilles de Gennes.

9:05AM M29.00006 A stochastic model for DNA electrotransfer with finite pulses , MIAO YU, HAO LIN, Rutgers University — Gene electrotransfer is a non-viral method to introduce foreign DNA into cells using electric fields. The fundamental mechanism for DNA transfer is unknown and under debate. While previous research investigated the role of DNA-membrane interaction and endocytosis, we here explore electrophoresis as a possible mechanism to assist translocation. In this model, DNA strands are treated as long-chain polymers driven through pores on the cell membrane by applied electric fields. A stochastic model is constructed, and solved numerically to parametrically study the time process of DNA translocation. Numerical results indicate that there exists an optimal pulse length beyond which DNA delivery probability no longer increases. The optimal length correlates inversely with applied field strength, and increases nonlinearly with DNA length. The results show good agreement with data from both solid-state nano-pore and electroporation experiments, and suggest that electrophoresis may play a key role in electroporation-mediated gene delivery.

9:18AM M29.00007 Dynamics of a microsphere in an anisotropic gel: a frontier in intracellular microrheology[1] , MANUEL GOMEZ-GONZALEZ, JUAN CARLOS DEL ALAMO, MAE Department, UC SanDiego — Particle tracking microrheology (PTM) is widely used to calculate the shear modulus of complex fluids. This technique is specially suitable to characterize the cell cytoplasm but its current formulation assumes isotropy, thereby rendering averaged shear moduli that are affected by the motion of the probe and do not represent accurately the inherent properties of the medium. This leads to errors greater than 100% when isotropic PTM is applied to live cells. In this work, we modify the PTM formulation to obtain the Directional Shear Moduli of an orthotropic medium. We study the motion of a spherical particle in an orthotropic medium, described by the Leslie-Ericksen equations. We calculate the drag force exerted on the particle, in the general case of 5 independent viscosity coefficients. In the idealized case where only 2 of the viscosity coefficients are important, we obtain a closed form analytical solution that allows us to modify current PTM formulae and obtain the directional shear moduli from the random thermal motion of a particle embedded in the cell cytoplasm.

[1]Supported by NSF CBET-1055697 CAREER Award.

9:31AM M29.00008 Flow instability and mixing of a complex fluid with extensional microstructure , MEREDITH BETTERTON, Department of Physics, University of Colorado, ANG-SHENG HANG, MICHAEL SHELLEY, Courant Institute, NYU — Several complex fluid systems show extensional dynamics in their microstructural evolution. Examples include isotropic-to-smectic liquid crystal phase transitions, and the relative sliding dynamics of microtubules induced by motor protein activity. To understand the macroscopic dynamics of such a system, we examine a simple kinetic model of a suspension of extending, hydrodynamically-coupled fibers. It is similar in structure to recent theories of motile suspensions. Our stability analysis shows the existence of large-scale flow instabilities, and numerical simulation of the nonlinear kinetic-fluid model shows the development of strongly time-dependent and complex macroscopic flows.

9:44AM M29.00009 Getting trapped in low-Re turbulence and collaborating your way out , ENKELEIDA LUSHI, Courant Institute NYU, and Imperial College London, MICHAEL J. SHELLEY, Courant Institute, New York University — We present an efficient numerical method to compute the dynamics of thousands of self-motile rod-like micro-swimmers that interact directly via the fluid flow they collectively generate. Using this method, we study the dynamics of rod-like swimmer particles in a background cellular vortical flow and show that hydrodynamic and steric interactions, the number of swimmers and system size, as well as the swimming mechanism ("Pusher" vs. "Puller"), have an effect on whether the swimmers get trapped in vortices or can escape them.

9:57AM M29.00010 Charge effect on solute transport across a periodic fiber array , MASAKO SUGIHARA-SEKI, TAKESHI AKINAGA, HIDEYUKI O-TANI, Kansai University — The luminal surface of vascular endothelial cells is covered by a fiber matrix layer referred to as the glycocalyx layer, and charge carried by the glycocalyx layer has been shown to significantly modulate the permeability of the microvessel wall to charged solutes. The present study is aimed to develop a fluid mechanical and electrostatic model for the transport of charged solutes across the glycocalyx layer and to examine the charge effect on the rate of diffusional and convectional transport of the solute. The glycocalyx layer was assumed to consist of identical circular cylinders with fixed surface charge, aligned parallel to each other in a hexagonal arrangement. For a spherical solute with fixed surface charge suspended in an electrolyte solution between circular cylinders, fluid mechanical and electrostatic analyses were carried out to calculate the flow field as well as the electric field around the solute to estimate the rate of solute transport across the layer. It was found that even at rather large ion concentrations, the repulsive electrostatic interaction between the solute and cylinder of like charge could significantly reduce the transport rate of the solute.

Tuesday, November 22, 2011 8:00AM - 10:10AM –
Session M30 Microfluids: Fluidic Devices II Ballroom IV

8:00AM M30.00001 Microfluidic Controlled Conformal Coating of Particles , SCOTT TSAI, School of Engineering and Applied Sciences, Harvard University, JASON WEXLER, JIANDI WAN, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University — Coating flows are an important class of fluid mechanics problems. Typically a substrate is coated with a moving continuous film, but it is also possible to consider coating of discrete objects. In particular, in applications involving coating of particles that are useful in drug delivery, the coatings act as drug-carrying vehicles, while in cell therapy a thin polymeric coating is required to protect the cells from the host's immune system. Although many functional capabilities have been developed for lab-on-a-chip devices, a technique for coating has not been demonstrated. We present a microfluidic platform developed to coat micron-size spheres with a thin aqueous layer by magnetically pulling the particles from the aqueous phase to the non-aqueous phase in a co-flow. Coating thickness can be adjusted by the average fluid speed and the number of beads encapsulated inside a single coat is tuned by the ratio of magnetic to interfacial forces acting on the beads.

8:13AM M30.00002 Self-assembling particles of various shapes and sizes into adjustable monolayers at liquid-fluid interfaces , SAI NUDURUPATI, MUHAMMAD JANJUA, Lake Superior State University, NADINE AUBRY, Carnegie Mellon University, PUSHPENDRA SINGH, New Jersey Institute of Technology — Capillarity induced self-assembly of particles trapped at a fluid-liquid interface has been widely used to fabricate self-assembled monolayers. Capillarity alone, however, cannot be used to form ultra-thin membranes, made of submicron or nanosized particles, or membranes with a non-close packed structure whose properties could be adjusted. To address these shortcomings, we apply an external electric field which generates electric field induced capillary forces which are non-negligible even on very small particles for which capillary forces based on their buoyant weight alone are nearly zero. In addition, particles experience repulsive electric forces which lead to expanded lattices whose constant can be adjusted by varying the strength and/or frequency of the applied electric field. In addition to forces, torques are involved in the case of non-spherical particles which then reorient themselves as they self-assemble.

8:26AM M30.00003 Electric field induced self-assembly of particles at liquid-liquid interfaces , M. HOSSAIN, B. DALAL, S. GURUPATHAM, I. FISCHER, P. SINGH, New Jersey Institute of Technology, N. AUBRY, Carnegie Mellon University — We have recently shown that virtually defect-free monolayers of particles can be assembled at air-liquid interfaces by applying an electric field normal to the interface. Particles rearrange themselves under the influence of interfacial and electrostatic forces to form hexagonal monolayers with long-range order. The lattice spacing can be varied by changing the electric field intensity and frequency. In this talk we present similar results for the self-assembly of particles on the interfaces between two liquids. It is shown that the monolayer arrangement depends on the radii of the particles, the dielectric properties of the fluids and particles, and the applied electric-field intensity.

8:39AM M30.00004 Capillary flow driven gradient generation in fluid stripes for biomaterial and biomedical applications , MATTHEW HANCOCK, Brigham & Women's Hospital, JIANKANG HE, FRANCESCO PIRAINO, GULDEN CAMCI-UNAL, ALI KHADEMHOSSEINI, Brigham & Women's Hospital — A simple and inexpensive bench-top method is presented employing passive mechanisms to generate centimeters-long gradients of molecules and particles in under a second with only a coated glass slide and a micropipette. By patterning hydrophilic regions on a substrate, a stripe of prepolymer solution is held in place on a glass slide by a hydrophobic boundary. Adding a droplet to one end of this "pre-wet" stripe causes a rapid capillary flow that spreads the droplet along the stripe to generate a gradient in the relative concentrations of the droplet and pre-wet solutions. Experiments and theoretical models characterize the flows and gradient profiles and their dependence on the fluid volumes, properties, and stripe geometry. A bench-top rapid prototyping method allows the user to design and fabricate the coated slides using only tape and hydrophobic spray. Gradient biomaterials are produced by crosslinking gradients of prepolymer solutions. Applications include producing a soluble drug gradient over cells in shear-protected microwells, generating a concentration gradient of cells encapsulated in three dimensions within a homogeneous biopolymer, and synthesizing a biomaterial with encapsulated cells exhibiting a gradient in cell spreading.

8:52AM M30.00005 Self-similar flow channel designs for parallel multiscale transport of multiple fluid species[1] , KENNETH LEE, OMER SAVAS, University of California, Berkeley — The need for multiscale fluid transport arises in a number of engineering applications involving fluid delivery or collection over a range of different lengthscales. A "tree-shaped" system of flow channels has been an efficient transport solution commonly practiced by biomimetics. There has been much work in optimizing these dendritic flow systems, primarily for cooling applications. However, most designs can be costly to manufacture and limited in scalability. Moreover, most systems are restricted to the transport of a single fluid species. This work explores the feasibility of self-similar flow channel designs to provide parallel multiscale transport of multiple fluid species. The self-similar characteristic of these designs simplifies manufacturing and allows for flexible scalability. Prototypes for the parallel transport of one and two independent fluid species are supported with analytical theory and experimental work. Designs for three and four species are presented as well.

[1]Supported by Joint Center for Artificial Photosynthesis (JCAP)

9:05AM M30.00006 Propulsion and Trapping of Micro-particles by Active Cilia Arrays , AMITABH BHATTACHARYA, Department of Chemical Engineering, University of Pittsburgh, GAVIN BUXTON, Department of Science, Robert Morris University, O. BERK USTA, The Center for Engineering in Medicine, 51 Blossom Street, Boston, MA 02114, USA, ANNA BALAZS, Department of Chemical Engineering, University of Pittsburgh — Hair-like cilia are vital for transporting fluid and particulates within and around biological organisms. While the coordinated motion of the cilia is effective at propelling the surrounding fluid, adhesive interaction between the cilia and particulates can be crucial for controlling particle movement. Here, we model transport of a microscopic particle via a regular array of beating elastic cilia, whose tips experience an adhesive interaction with the particle's surface. At optimal adhesion strength, the average particle velocity is maximized. Using simulations spanning a range of cilia stiffness, particle radius, and cilia-particle adhesion strength, we explore the parameter space over which the particle can be "released," "propelled" or "trapped" by the cilia. We use a semi-analytical theory to predict parameters for which the cilia are able to attach themselves to the particle. This is the first study that shows how both stiffness and adhesion strength are crucial for the manipulation of particles by active cilia arrays. These results can ultimately facilitate the design of synthetic cilia that integrate adhesive and hydrodynamic interactions to selectively trap or repel particulates.

9:18AM M30.00007 Experiment and Modeling of Spatially Indexed Microbead Arrays for High-Throughput Screening Applications , THOMAS LEARY, CHARLES MALDARELLI, ALEXANDER COUZIS, City College of New York — The development of platforms for multiplexed, high throughput screening of the binding interactions of target biomolecules against a library of potential binding probes enables progress in many areas in medicine and biology. Formats in which probes are linked to microbeads arrayed in a microfluidic channel offer high sensitivity, reduced reagent consumption and are easily parallelized for multiplexed detection. This presentation describes a microfluidically assembled microbead array in which beads are streamed through a channel with an array of wells inscribed in the floor of the channel. The beads are captured in the wells via gravity. We demonstrate that an array of beads displaying different receptors can be assembled in this format, indexed by sequential depostion and used for a prototype assay. Solutions for a two dimensional mass transfer model of the conjugation of the probe to the receptor on the bead surface identify kinetically limited regimes which are used to measure the binding kinetics of the prototype assay.

9:31AM M30.00008 Isotachophoretic Preconcentration of Cardiac Proteins from Human Serum , PRASHANTA DUTTA, MOHAMMAD HOSSAN, TALUKDER JUBERY, DANNY BOTTENUS, CORNELIUS IVORY, Washington State University — Cationic isotachophoresis (ITP) is used to concentrate and detect a cardiac biomarker, cardiac troponin I (cTnI), spiked into depleted human serum in a cascade poly(methyl methacrylate) (PMMA) microfluidic channel. PMMA microchannel was formed using solvent imprinting and temperature-assisted bonding. A 100x reduction in cross-sectional area is implemented by gradually decreasing the channel width and height to increase the sensitivity of ITP. The ITP was performed in peak mode with potassium ion as the leading electrolyte and hydronium ions as the terminating electrolyte. The cross-sectional area reductions in combination with ITP allowed visualization of lower concentrations of fluorescently labeled cTnI. Experimental results show that ITP can detect cTnI at initial concentrations as low as 46 ng/mL in the presence of human serum proteins, and ITP can obtain cTnI concentrations factors as high as 10,000. This demonstrates the detection of cTnI in depleted human serum at clinically relevant concentrations without the use of antibodies and relying solely on ITP in a cascade microchip.

9:44AM M30.00009 Acoustically Generated Flows in Flexural Plate Wave Sensors: a Multifield Analysis , ERSIN SAYAR, BAKHTIER FAROUK, Department of Mechanical Engineering and Mechanics, Drexel University — Acoustically excited flows in a microchannel flexural plate wave device are explored numerically with a coupled solid-fluid mechanics model. The device can be exploited to integrate micropumps with microfluidic chips. A comprehensive understanding of the device requires the development of coupled two or three-dimensional fluid structure interactive (FSI) models. The channel walls are composed of layers of ZnO, Si_3N_4 and Al. An isothermal equation of state for the fluid (water) is employed. The flexural motions of the channel walls and the resulting flowfields are solved simultaneously. A parametric analysis is performed by varying the values of the driving frequency, voltage of the electrical signal and the channel height. The time averaged axial velocity is found to be proportional to the square of the wave amplitude. The present approach is superior to the method of successive approximations where the solid-liquid coupling is weak.

9:57AM M30.00010 Electric field-enhanced water dissociation in nanoporous membranes[1] , DEVIN CONROY, RICHARD CRASTER, OMAR MATAR, Imperial College London, HSUEH-CHIA CHANG, Notre Dame University — The dissociation of water with a PN junction is investigated theoretically. The model accounts for the the voltage potential and transport of ions using the Nernst-plank and Maxwell's equations respectively and solved numerically and asymptotically in the small reaction limit. The advantage of the PN junction is that the electric field is very large, which enhances the product yield far from the thermodynamic value. Further, the junction causes the charge of the ions to be separated onto the oppositely charged membrane, forming a travelling wave the has a constant velocity. Preliminary work indicates that the travelling wave undergoes a mobility gradient driven instability, for sufficiently large electric fields, that leads to the formation of fingers.

[1]Royal Academy of Engineering and EPSRC (EP/E046029/1)

Tuesday, November 22, 2011 10:40AM - 11:15AM –

Session N27 Invited Session: Capillary Phenomena at Nanoscales: Electrowetting and Capillary Adhesion Ballroom I-II

10:40AM N27.00001 Capillary Phenomena at Nanoscales: Electrowetting and Capillary Adhesion[1] , MARK ROBBINS, Johns Hopkins University — Theories of capillary phenomena have traditionally been based on continuum approximations that break down as dimensions shrink to nanometer scales. Molecular simulations are used to test the limits of continuum theory in electrowetting on dielectric (EWOD) and capillary adhesion between solids. In EWOD, a fluid drop is separated from an electrode by a dielectric. Increasing the voltage V between drop and electrode, decreases the contact angle θ, allowing the droplet to be manipulated. Simulations of nanoscale drops show the same behavior as experiments on millimeter drops. The contact angle follows the continuum Young-Lippmann equation (YLE) at low voltages and then saturates. The saturation mechanism has been difficult to identify in experiments. Simulations show that charged molecules are pulled from the drop by large electrostatic forces near the contact line. Saturation can be delayed by increasing molecular binding, lowering temperature or increasing dielectric constant. A local force balance equation is derived that agrees with the YLE below saturation and remains valid after saturation. Simulations of capillary adhesion examined the force between a spherical tip of radius R and a flat substrate. The shape of the meniscus agrees remarkably well with continuum theory down to nanometer separations, as does the adhesive force from interfacial tension. However, the total force may deviate by factors of two or have the opposite sign. While the component of the pressure along the substrate agrees with the Laplace pressure from continuum theory, the out-of –plane component does not. There may also be significant force oscillations associated with layering near the solids. The elastic response of the solid has little affect on adhesive forces.

[1]This material is based upon work supported by National Science Foundation Grant No. CMS-0103408

Tuesday, November 22, 2011 10:40AM - 11:15AM –

Session N29 Invited Session: Three little stories on Fragmentation and Mixing Ballroom III-IV

10:40AM N29.00001 Three little stories on Fragmentation and Mixing , EMMANUEL VILLERMAUX, Aix Marseille Universite — Mixing, the process by which objects aggregate and lose their identity as they merge into a homogeneous ensemble and Fragmentation, the process by which initially connected objects disjoin, seem at first sight to pertain to different worlds. Several observations suggest on the contrary the profound unity between these two phenomena.

Tuesday, November 22, 2011 11:20AM - 11:40AM –

Session P27 Andreas Acrivos Dissertation Award Lecture Ballroom I-II

11:20AM P27.00001 Andreas Acrivos Dissertation Prize Lecture: Stability of inviscid flows from bifurcation diagrams exploiting a variational argument[1] , PAOLO LUZZATTO-FEGIZ, Woods Hole Oceanographic Institution — Steady fluid solutions play a special role in the dynamics of a flow: stable states may be realized in practice, while unstable ones may act as attractors. Unfortunately, determining stability is often a process far more laborious than finding steady states; indeed, even for simple vortex or wave flows, stability properties have often been the subject of debate. We consider here a stability idea originating with Lord Kelvin (1876), which involves using the second variation of the energy, $\delta^2 E$, to establish bounds on a perturbation. However, for numerically obtained flows, computing $\delta^2 E$ explicitly is often not feasible. To circumvent this issue, Saffman & Szeto (1980) proposed an argument linking changes in $\delta^2 E$ to turning points in a bifurcation diagram, for families of steady flows. Later work has shown that this argument is unreliable; the two key issues are associated with the absence of a formal turning-point theory, and with the inability to detect bifurcations (Dritschel 1995, and references therein). In this work, we build on ideas from bifurcation theory, and link turning points in a velocity-impulse diagram to changes in $\delta^2 E$; in addition, this diagram delivers the direction of the change of $\delta^2 E$, thereby providing information as to whether stability is gained or lost. To detect hidden solution branches, we introduce to these fluid problems concepts from imperfection theory. The resulting approach, involving "imperfect velocity-impulse" diagrams, leads us to new and surprising results for a wide range of fundamental vortex and wave flows; we mention here the calculation of the first steady vortices without any symmetry, and the uncovering of the complete solution structure for vortex pairs. In addition, we find precise agreement with available results from linear stability analysis.

[1]Doctoral work advised by C.H.K. Williamson at Cornell University.

Tuesday, November 22, 2011 11:20AM - 11:40AM –

Session P29 Francois Frenkiel Best Paper Award Lecture Ballroom III-IV

11:20AM P29.00001 Francois Frenkiel Award Lecture: Low-Order Representations of Irregular Surface Roughness and Their Impact on a Turbulent Boundary Layer , K.T. CHRISTENSEN[1], University 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 highly-irregular surface roughness replicated from a turbine blade damaged by deposition of foreign materials were generated using singular value decomposition (SVD) to decompose the complex topography into a set of topographical basis functions of decreasing importance to the original ("full") surface character. The low-order surface models were then formed by truncating the full set of basis functions at the first 5 and 16 modes (containing approximately 71% and 95% of the full surface content, respectively), so that only the most dominant, and large-scale, topographical features were included in the models. Physical replications of the full surface and the two low-order models were created using rapid prototyping methods and particle image velocimetry was used to acquire ensembles of instantaneous velocity fields in the streamwise-wall-normal plane and a streamwise-spanwise plane deep within the roughness sublayer at moderate Reynolds number. The efficacy of the low-order surface models in replicating the statistical and structural characteristics of the full-surface flow will be highlighted, particularly within the roughness sublayer where the flow behavior is tightly coupled to the roughness features.

[1]In collaboration with R. Mejia-Alvarez, University of Illinois.

Tuesday, November 22, 2011 12:50PM - 2:34PM –

Session R1 Geophysical Flows: General II 301

12:50PM R1.00001 Evolution of shallow, horizontal shear layers with a horizontal density contrast[1] , KARL HELFRICH, Woods Hole Oceanographic Institution, BRIAN WHITE, University of North Carolina, Chapel Hill, ANA KARINA RAMOS-MUSALEM, National Autonomous University of Mexico — Shallow coastal ocean flows frequently involve strong horizontal shear layers in combination with a horizontal density gradient. In the absence of the density contrast, the flow undergoes the classic Rayleigh instability leading to the roll-up of the shear layer into vertical vortices. The density contrast results in a transverse gravity-driven tilting of the interface resembling a lock-exchange. The evolution of this rapid buoyancy-induced tilting of horizontal shear is explored with laboratory experiments performed in a new open-channel flume with a flapping, splitter-plate entrance. Measurements of the downstream evolution are made with co-incident PIV and LIF in horizontal planes at several vertical locations spanning the water column. The measurements show vortex roll-up and tilting and the subsequent emergence of horizontal Kelvin-Helmholtz billows that form on the interface and interact with the primary vortices. The characteristics of flow are discussed, including phase averaged and mean velocity, vorticity and density fields as a function of a scaling parameter that quantifies the relative effects of lateral shear and buoyancy adjustment. The experiments compare favorably with three-dimensional, implicit-LES, numerical model solutions for the experimental configuration and parameters.

[1]Supported by NSF grant OCE-0726161.

1:03PM R1.00002 Self-similar structure of a corner wave at short times[1] , PABLO MARTINEZ-LEGAZPI, JAVIER RODRIGUEZ-RODRIGUEZ, Carlos III Univ. of Madrid, Spain, JUAN C. LASHERAS, UC San Diego — We study theoretically the flow near the corner of a vertical flat plate partially submerged across an uniform stream. When the Froude number is large enough, a three dimensional wave forms at the corner of the plate which evolves downstream in a similar way as a time-evolving two dimensional plunging wave. We have performed pressure-impulse asymptotic analysis of the flow near the origin of the corner wave to describe the initial evolution of the wave and to clarify the physical mechanism that leads to its formation. The analysis shows that the wave crest exhibits a self similar behavior at short times. After this self-similar stage, the wave crest detaches plunges following a nearly ballistic trajectory. The results improve our computational modeling of the flow near the stern of a high-speed surface ship, providing the initial condition needed for CFD simulations to properly capture the behavior of these stern waves.

[1]Supported by the ONR through grant N00014-1110508.

1:16PM R1.00003 Breaking Wave Impact on a Partially Submerged Rigid Cube in Deep Water[1]

, C.M. IKEDA, University of Maryland, M. CHOQUETTE, Carnegie Mellon University, J.H. DUNCAN, University of Maryland — The impact of a plunging breaking wave on a partially submerged cube is studied experimentally. The experiments are performed in a wave tank that is 14.8 m long, 1.15 m wide and 2.2 m high with a water depth of 0.91 m. A single repeatable plunging breaker is generated from a dispersively focused wave packet (average frequency of 1.4 Hz) that is created with a programmable wave maker. The rigid ($L = 30.5$ cm) cube is centered in the width of the tank and mounted from above with one face oriented normal to the oncoming wave. The position of the center of the front face of the cube is varied from the breaker location ($x_b \approx 6.35$ m) to $x_b + 0.05$ m in the streamwise direction and from $-0.25L$ to $0.25L$ vertically relative to the mean water level. A high-speed digital camera is used to record both white-light and laser-induced fluorescence (LIF) movies of the free surface shape in front of the cube before and after the wave impact. When the wave hits the cube just as the plunging jet is formed, a high-velocity vertical jet is created and the trajectory and maximum height of the jet are strongly influenced by the vertical position of the cube.

[1]Supported by the Office of Naval Research, Contract Monitor R. D. Joslin.

1:29PM R1.00004 Simulation-based study of wind load on surface-piercing body and its dependency on waves

, SHENGBAI XIE, LIAN SHEN, Johns Hopkins University — The wind load acting on surface ships and offshore structures are important for their operation and safety. In this study, we simulate the flow fields of wind and wave past a surface-piercing structure using a multi-scale modeling strategy. At large scales, the turbulent wind is simulated using large-eddy simulation on boundary-fitted grid coupled with nonlinear wavefield simulation using a high-order spectral method. The large-scale simulation provides environmental inputs for the local-scale simulation around the body. At local scales, the air and water flows are simulated by a coupled level-set and volume-of-fluid method. An immersed boundary method is used to represent the body. From the simulation, the statistics and structure of the wind and wave fields around the body are elucidated, and the loads on the body are quantified. A variety of developing and fully developed wave spectra at different sea states are considered. Swells and their interactions with the local wind-waves are included in the simulation. It is found that the mean wind loads are highly dependent on the wave conditions, and the instantaneous wind forcing varies with the wave phase.

1:42PM R1.00005 Simulation of flow past a sphere in a stratified fluid

, MATTHEW DE STADLER, SUTANU SARKAR, University of California San Diego — Direct numerical simulation is used to simulate spatially-evolving flow past a sphere in a stratified fluid. The immersed boundary method is used to treat the sphere inside the domain. The main objective of this study is to characterize the near wake region. Statistics of interest include the drag coefficient, separation angle, Strouhal number, and the spatial evolution of the velocity fluctuations and the defect velocity. In addition to quantitative statistics, visualizations of the vortex structures in the wake will also be provided and discussed. Results are compared and contrasted with previous experimental and numerical data for unstratified and stratified flow past a sphere.

1:55PM R1.00006 Falling bodies through sharply stratified fluids: theory and experiments[1]

, RICHARD MCLAUGHLIN, ROBERTO CAMASSA, University of North Carolina, CLAUDIA FALCON, STEVE HARENBERG, KEITH MERTENS, JOHNNY REIS, WILLIAM SCHLIEPER, BAILEY WATSON, BRIAN WHITE, University of North Carolina, UNC RTG FLUIDS GROUP TEAM — The motion of bodies and fluids moving through a stratified background fluid arises naturally in the context of carbon (marine snow) settling in the ocean, as well as less naturally in the context of the DWH Gulf oil spill. The details of the settling rates may affect the ocean contribution to the earth's carbon cycle. We look at phenomena associated with many falling spheres in stratified fluids, as well as behavior of multiphase buoyant plumes penetrating strong stratification. We present careful measurements critical heights for fully miscible jets and companion analytical prediction. In turn, we examine cases involving clouds of sinking particulate and rising buoyant oil emulsions and associated plume trapping behaviors.

[1]NSF DMS RTG 0943851, NSF DMS 1009750, NSF CMG ARC-1025523, NSF RAPID CBET-1045653

2:08PM R1.00007 Vortex rings passing through sharp density transitions[1]

, KEITH MERTENS, ROBERTO CAMASSA, RICHARD MCLAUGHLIN, ELAINE MONBUREAU, DAVID NENON, CASEY SMITH, CLAUDIO VIOTTI, BRIAN WHITE, University of North Carolina, Chapel Hill — In this presentation the various behaviors of a dense miscible vortex ring descending through a sharp stratification are explored. By varying experimental parameters it is found the ring dynamics can transition from those which pass through the interface to those which can become totally entrapped within the density transition, even though in all cases the ring is the most dense fluid in the system. Experimental results will outline these various behaviors and quantify this transition with an experimental phase diagram. A scaling law will be derived from dimensional considerations to predict this transition in behavior. Numerical simulations will also be presented and compared to the previous experimental results.

[1]NSF DMS RTG 0943851, NSF DMS 1009750, NSF CMG ARC-1025523, NSF RAPID CBET-1045653

2:21PM R1.00008 A comparison between laboratory and numerical simulations of gravity-driven coastal currents with a geostrophic theory

, SANDY GREGORIO[1], PETER THOMAS, The University of Warwick, DALE HAIDVOGEL, Rutgers University, EZGI TASKINOGLU, SRA International, ANDREW SKEEN, Etalon Research Ltd. — Laboratory and numerical simulations of buoyant, gravity-driven coastal currents are summarized and compared to the inviscid geostrophic theory of Thomas & Linden 2007.[2] The lengths, widths and velocities of the buoyant currents are studied. Agreement between the laboratory and numerical experiments and the geostrophic theory is found to depend on two non-dimensional parameters which characterize, respectively, the steepness of the plumes isopycnal interface and the strength of horizontal viscous forces (quantified by the horizontal Ekman number). The best agreement between experiments (both laboratory and numerical) and the geostrophic theory are found for the least viscous flows. At elevated values of the horizontal Ekman number, laboratory and numerical experiments depart more significantly from theory.

[1]MEOM/LEGI from the 1st of October 2011

[2]Thomas, P. J. and Linden, P.F. 2007. Rotating gravity currents: small-scale and large-scale laboratory experiments and a geostrophic model. *J. Fluid Mech.* **578**, 35-65.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R2 Granular Flows IV 302

12:50PM R2.00001 Correlating Grain and Slider Dynamics in a Stick-Slip Experiment[1] , ERIC SIA, BOB BEHRINGER, Duke University — We describe an experiment to characterize the stick-slip nature of sliding friction. In the experiment a solid slider is pulled by a spring moving at constant velocity across a 2D granular bed of bidisperse photoelastic discs confined to a vertical channel. A force sensor attached to the spring measures the pulling force on the slider, while two accelerometers on the slider provide angle and acceleration data. Synchronous photoelastic images of the granular bed acquired by a camera moving along with the slider allow us to correlate all features of the slider and granular dynamics. We particularly focus on the correlation of the slider acceleration and changes in the granular force network. This experiment can help us better understand both atomic scale friction experiments, and stick-slip at the geophysical scale.

[1]NSF-DMR0906908, ARO-W911NF-11-1-0110

1:03PM R2.00002 Flowing layer thickness in a granular tumbler , RICHARD M. LUEPTOW, FLORENT PIGNATEL, Northwestern University, CAROLINE ASSELIN, LUCAS KRIEGER, French Air Force Academy, JULIO M. OTTINO, Northwestern University — The thickness of the flowing layer in a tumbler varies substantially depending on flow conditions, but no predictive approach is available. We have studied monodisperse granular flows in air, water, and glycerine in a quasi-2D tumbler, focusing on the rolling regime to highlight common features and differences between dry and wet granular flows. For dry granular flows, the flowing layer thickness measured in units of particle diameter scales with a dimensionless flow rate based on the tumbler rotational speed and radius as well as the particle diameter. Using appropriate characteristic time scales, the data for the dry and liquid experiments also collapse fairly well. The Stokes number is a key dimensionless parameter to characterize the flow of granular material in liquids.

1:16PM R2.00003 Anatomy of flow and jam of a two dimensional granular flow in a hopper[1] , JUNYAO TANG, ROBERT BEHRINGER, PROCTER & GAMBLE COLLABORATION[2] — We seek an understanding of the physics of jamming for hopper flow using high speed spatio-temporal video data for photoelastic disks flowing through a two-dimensional hopper. We have found experimental support for the hypothesis that jamming events of granular flow in a hopper occur as a Poisson process. Through measuring the density field and stress distribution, we demonstrate that "dome-structured" stress chains affect the density and velocity fields, leading to the mean flow profile in the hopper. We also measure how jamming statistics depend on the orifice sizes and the wall angles of a hopper. By calculating stress fluctuations, we conclude that instead of the density field, the formation of localized force chain arches controls the jamming transition of granular flow in a hopper. These data are part of an IFPRI-NSF Collaboratory for comparing physical data and simulations.

[1]This work is funded by IFPRI (International Fine Particle Research Institute).
[2]Paul Mort

1:29PM R2.00004 Anisotropic surface friction , STEPHAN KOEHLER, IBAR DELACRUZ, NATHAN NESBITT, IAN MORSE, Physics, WPI — Contrary to the literature, we find that altering the surface roughness has a large effect on intruder drag in the quasi-static regime. Moreover we pattern the surface with a sawtooth texture and observe anisotropic drag: when the texture is comparable in size to the bead diameter the frictional force is 1/3rd greater for the flow directed against the sawtooth versus the opposite flow. We present a systematic study showing the dependence of the anisotropy on the texture's orientation.

1:42PM R2.00005 Microstructural theory for colloidal suspensions in active microrheology , EHSSAN NAZOCKDAST, JEFFREY MORRIS, Levich Institute & Chemical Engineering of CUNY — The active microrheology problem where a probe particle is pulled through a bath of colloidal particles with a constant external force F^{ext} is studied using a theoretical framework based on the Smoluchowski equation, with emphasis on concentrated colloidal suspensions far from equilibrium. The probability distribution of bath particles with respect to the probe, $g(r)$, is determined from an integro-differential equation solved by iterative numerical methods at different set of Pe (ratio of external to Brownian forces) and volume fractions. The role of inter-particle interactions on microstructure is studied by using different types of pair potentials with the intention to model systems ranging from hard-spheres to soft colloids. The obtained distribution is then used to compute the apparent shear viscosity. The predictions of microstructure and rheology are compared with our Accelerated Stokesian Dynamics simulations and available experimental results.

1:55PM R2.00006 Effect of particle devolatilization on bed dynamics during biomass thermochemical conversion , PERRINE PEPIOT, JESSE CAPECELATRO, OLIVIER DESJARDINS, Cornell University — Fluidization is a technique of choice for the thermochemical conversion of biomass. At conversion temperatures however, the amount of gases released by the biomass is large enough to impact the mixing of the reactive particles with the inert sand, and modify the bubbling frequency and intensity. This, in turn, may significantly affect the chemical processes and the final product distribution. In this context, optimizing reactor design and operating conditions requires a better understanding of the actual bed dynamics in the presence of reactive particles. In this work, two- and three-dimensional simulations of biomass conversion in a lab-scale fluidized bed reactor are conducted using a Lagrangian approach to handle the solid phase. The biomass devolatilization chemistry is described using a commonly used global model taking into account each constituent of the biomass. Statistical analysis of the particles and velocity fields is conducted and results are compared to non-reactive cases to quantify the effect of devolatilization on particle mixing, especially segregation, and on the bubbling pattern of the bed.

2:08PM R2.00007 Surface Chemistry Effects in Submerged Granular Flows of Hydrophobic Grains , BRIAN UTTER, BENJAMIN FOLTZ, James Madison University — We experimentally investigate submerged granular flows of hydrophobic and hydrophilic grains in a rotating drum. While slurry and suspension flows are common in nature and industry, effects of surface chemistry on flow behavior have received relatively little attention. The experiment consists of a cylindrical drum containing various concentrations of hydrophobic and hydrophilic grains of sand submerged in water rotated at a constant angular velocity. Sequential images of the resulting avalanches are taken and analyzed. While it is known that at slow speeds, submerged avalanches appear qualitatively similar to dry flows, our results suggest that the surface properties of the grains affect underwater flow significantly. High concentrations of hydrophobic grains result in an effectively cohesive interaction between the grains forming aggregates. We present data on the size of the aggregates, slope, and avalanche statistics with changes in concentration. The formation and nature of the aggregates depends significantly on the presence of air in the system. At concentrations larger than 75% hydrophobic sand, the avalanches do not behave in a manner which is typical for sand, but as the concentration decreases, the aggregates are smaller, the angle of repose decreases, and the grains start showing rheological properties similar to those in regular sand.

2:21PM R2.00008 Percolation theory applied to the force field in granular material[1] , ARNAUD GOULLET, LOU KONDIC, New Jersey Institute of Technology — We analyze the structure of the force field in slowly compressed granular system by the means of discrete element simulations. Using the tools of percolation theory, we compute the quantities describing the force field and discuss their dependence on polydispersity and frictional properties of the granular particles. Then, we correlate the results to the ones obtained using topological approach based on Betti numbers which measure the number of clusters as a function of force thresholds.

[1]Supported by NSF Grant No. DMS-0835611

2:34PM R2.00009 Material point method simulation of dense granular material , DUAN ZHANG, XIA MA, Los Alamos National Laboratory — Accurate modeling and simulation of granular flow or deformation require a numerical method with Lagrangian description to account for history dependence of the material. However, large deformation or flow of the material requires an Eulerian description. Numerically, different descriptions of the material result in different codes and applications. Unsatisfactory results have been reported by many modelers using both methods. For instance, element deletion scheme is used in the finite element method, a Lagrangian description, to eliminate the highly distorted elements, which results in artificial reduction of inertia from the problem. In codes using the finite volume method, an Eulerian descriptions, how to advect brittle damage of the material is a significant issue. To address these issues we use the material point method, which uses Lagrangian material points and Eulerian mesh simultaneously. Improvements are made to the original material point method for our applications. It is found that the improvements are critically important to granular flows results from brittle damage of the material, while it is marginally important to ductile materials.

2:47PM R2.00010 Couette and Fourier planar ows for a low density granular gas , FRANCISCO VEGA REYES, VICENTE GARZÓ, ANDRÉS SANTOS, Departamento de Física, Universidad de Extremadura, Spain — We study in this work steady laminar flows in a low density granular gas. Our system is excited by shear and temperature sources at the boundaries (two infinite parallel walls). We describe previously unreported types of non-Newtonian granular flows. We obtain also their corresponding rheologic and hydrodynamic transport coefficients, following three independent methods: 1) an analytical solution, obtained from Grad's method applied to the inelastic Boltzmann equation, 2) a numerical solution of the inelastic Boltzmann equation, obtained by means of the Direct Simulation Monte Carlo method, and 3) molecular dynamics simulations. We show that the three procedures yield the same general classification of planar Fourier and Couette flows for the granular gas.

Tuesday, November 22, 2011 12:50PM - 3:00PM – Session R3 Bubbles IV 303

12:50PM R3.00001 Velocimetry in cavitating flows by X-ray imaging , OLIVIER COUTIER-DELGOSHA, LML Laboratory / Arts et Metiers ParisTech, MARKO HOCEVAR, Laboratory for Water and Turbine Machines, Faculty of Mechanical Engineering, University of Ljubljana, ILYASS KHLIFA, Arts et Metiers ParisTech / LML LaboratoArts et Metiers ParisTech / LML Laboratoryry, SYLVIE FUZIER, Arts et Metiers ParisTech / LML Laboratory, ALEXANDRE VABRE, CEA LIST, KAMEL FEZZAA, WAH-KEAT LEE, X-ray Science Division, Argonne National Laboratory, LABORATORY FOR WATER AND TURBINE MACHINES, FACULTY OF MECHANICAL ENGINEERING, UNIVERSITY OF LJUBLJAN COLLABORATION, CEA LIST COLLABORATION, X-RAY SCIENCE DIVISION, ARGONNE NATIONAL LABORATORY COLLABORATION — A promising method to measure velocity fields in a cavitating flow is presented. Dynamics of the liquid phase and of the bubbles are both investigated. The measurements are based on ultra fast X-ray imaging performed at the APS (Advanced Photon Source) of the Argonne National Laboratory. The experimental device consists of a millimetric Venturi test section associated with a transportable hydraulic loop. Various configurations of velocity, pressure, and temperature have been investigated. The slip velocity between vapor and liquid is calculated everywhere both velocities can be obtained. Reynolds stresses are also calculated, and compared with the ones obtained in non-cavitating conditions.

1:03PM R3.00002 Influence of Vortex Cavitation on Vortex Breakdown , HARISH GANESH, ANDRE PINHEIRO, University of Michigan, Ann Arbor, BU GUEN PAIK, Maritme & Ocean Engineering Research Institute/KORDI, Daejeon, Korea, STEVEN L. CECCIO, University of Michigan, Ann Arbor — Cavitation inception and development was observed in the vortices created by an $\Lambda = 70°$ delta wing over a range of attack angles and cavitation numbers. The location of cavitation inception, bubble size, growth rate, and the effect of cavitation on the location of vortex breakdown were studied. From the observations a rationale that governs the observed inception location, growth of incepted nucleus into a given shape is proposed. It is based on the alterations of the vortical core flow that an elongating bubble will cause. The minimizing potential theory of *Rusak et al*[1] is used to identify the constraints on the bubble growth which in turn helps us to understand the bubble shape and growth rate.

[1] "The evolution of a perturbed vortex in a pipe to axisymmetric vortex breakdown," Rusak, Z., Wang. S., Whiting. C. H., *Journal of fluid mechanics*, 1998, Vol 336, pp 211-237

1:16PM R3.00003 High-speed microjet generation using laser-induced vapor bubbles , NIKOLAI OUDALOV, YOSHIYUKI TAGAWA, IVO PETERS, CLAAS-WILLEM VISSER, DEVARAJ VAN DER MEER, ANDREA PROSPERETTI, CHAO SUN, DETLEF LOHSE, University of Twente — The generation and evolution of microjets are studied both experimentally and numerically. The jets are generated by focusing a laser pulse into a microscopic capillary tube ($\sim$50 μm) filled with water-based red dye. A vapor bubble is created instantly after shooting the laser ($<$1 μs), sending out a shockwave towards the curved free surface at which the high-speed microjet forms. The process of jet formation is captured using high-speed recordings at 1.0×10^6 fps. The velocity of the microjets can reach speeds of $\sim$850 m/s while maintaining a very sharp geometry. The high-speed recordings enable us to study the effect of several parameters on the jet velocity, e.g. the absorbed energy and the distance between the laser spot and the free surface.The results show a clear dependence on these variables, even for supersonic speeds. Comparisons with numerical simulations confirm the nature of these dependencies.

1:29PM R3.00004 Bubbles in drops: from cavitation to exploding stars[1] , PHILIPPE KOBEL, DANAIL OBRESCHKOW, NICOLAS DORSAZ, AURELE DE BOSSET, MARC TINGUELY, MOHAMED FARHAT, Ecole Polytechnique Federale de Lausanne — We performed an experiment to generate single cavitation bubbles inside centimetric quasi-spherical water drops. To produce such drops, our experiment was realized under microgravity conditions (42nd ESA parabolic flight campaign). The ultra-fast recording of the bubble collapse and ensuing dynamics revealed consequences of the unique geometry of the drop's free surface. We obtained the first visualizations of a jet pair escaping the drop after the collapse of eccentrically-placed bubbles. The high quality of the images also disclosed some features of the inner drop dynamics. Due to their confinement within the isolated drop volume, shock waves emitted at the bubble collapse bounce back and forth thereby exciting gas nuclei into sub-millimetric bubbles. When located beneath the free surface, the collapse of these bubbles gives rise to narrow "hair-like" jets on the surface. Here we briefly describe the physics underlying these observations while discussing possible analogies with various astrophysical processes from the Sun (spicules) to asymmetric supernovae.

[1]SNSF Grant 200020-116641

1:42PM R3.00005 Homogeneous cavitation in microfluidics: a record high dynamic tensile threshold , KEITA ANDO, AI-QUN LIU, CLAUS-DIETER OHL, Nanyang Technological University — An experimental technique is presented which allows one to measure the rupture strength of water using a microfluidic approach. A transparent microfluidic channel is filled with clean water, partially leaving an air-water interface. A focused infrared laser pulse within the liquid creates a spherical shock wave near the interface. The shock reflects, due to acoustic impedance mismatch, as a strong tension wave with high negative pressures. The liquid becomes stretched and at the homogeneous cavitation threshold ruptures with the nucleation of vapor bubbles. These bubbles are captured using an optical delay and very short exposure times. Reproducible observations of the bubble nucleation are obtained, supporting our claim of homogeneous bubble nucleation. Multicomponent Euler flow simulation estimates a tensile stress threshold of -60 MPa, which is the largest reported tension for dynamic measurements.

1:55PM R3.00006 Understanding Cavitation Intensity through Pitting and Pressure Pulse Analysis

, A. JAYAPRAKASH, S. SINGH, J-K. CHOI, G. CHAHINE, Dynaflow Inc. — Cavitation erosion is of interest to the designers of ship propulsion devices because of its detrimental effects. One of the difficulties of predicting cavitation erosion is that the intensity of cavitation is not well predicted or defined. In this work we attempt to define the intensity of a cavitation erosion field through analysis of cavitation induced erosion pits and pressure pulses. In the pitting tests, material samples were subjected to cavitation field for a short duration of time selected within the test sample's incubation period, so that the test sample undergoes plastic deformation only. The sample material reacts to these cavitation events by undergoing localized permanent deformation, called pits. The resulting pitted sample surfaces were then optically scanned and analyzed. The pressure signals under cavitating jets and ultrasonic horns, for different conditions, were experimentally recorded using high frequency response pressure transducers. From the analysis of the pitting data and recorded pressure signals, we propose a model that describes the statistics, which in the future can be used to define the cavitation field intensity. Support for this work was provided by Office of Naval Research (ONR) under contract number N00014-08-C-0450, monitored by Dr. Ki-Han Kim.

2:08PM R3.00007 Vapor bubble evolution on a heated surface containing open microchannels[1]

, CHRISTOPHER J. FORSTER, ARI GLEZER, MARC K. SMITH, Georgia Institute of Technology — Power electronics require cooling technologies capable of high heat fluxes at or below the operating temperatures of these devices. Boiling heat transfer is an effective choice for such cooling, but it is limited by the critical heat flux (CHF), which is typically near 125 W/cm^2 for pool boiling of water on a flat plate at standard pressure and gravity. One method of increasing CHF is to incorporate an array of microchannels into the heated surface. Microchannels have been experimentally shown to improve CHF, and the goal of this study is to determine the primary mechanisms associated with the microchannels that allow for the increased CHF. While the use of various microstructures is not new, the emphasis of previous work has been on heat transfer aspects, as opposed to the fluid dynamics inside and in the vicinity of the microchannels. This work considers the non-isothermal fluid motion during bubble growth and departure by varying channel geometry, spacing, and heat flux input using a level-set method including vaporization and condensation. These results and the study of the underlying mechanisms will aid in the design optimization of microchannel-based cooling devices.

[1]Supported by ONR.

2:21PM R3.00008 Acoustically enhanced boiling heat transfer on a heated surface containing open microchannels[1]

, THOMAS R. BOZIUK, MARC K. SMITH, ARI GLEZER, Georgia Institute of Technology — Acoustic actuation is used to enhance boiling heat transfer on a submerged heated surface containing an array of open microchannels by controlling the formation and evolution of vapor bubbles and inhibiting the instability that leads to film boiling at the critical heat flux. The effect of actuation at millimeter and micrometer scales is investigated with emphasis on the behavior of bubble nucleation, growth, contact-line motion, condensation, and detachment. The results show that microchannels control the location of boiling and reduce the mean surface superheat. In addition, acoustic actuation increases the heat flux at a given surface temperature and leads to a significant increase in the critical heat flux, a reduction of the vapor mass above the surface, and the breakup of low-frequency vapor slug formation.

[1]Supported by ONR.

2:34PM R3.00009 The most spherical cavitation bubble[1]

, MARC TINGUELY, DANAIL OBRESCHKOW, PHILIPPE KOBEL, NICOLAS DORSAZ, AURELE DE BOSSET, MOHAMED FARHAT, Ecole Polytechnique Federale de Lausanne — Focusing a high energy pulsed laser into water is a widely used method to generate a single cavitation bubble. Such a bubble is generally assumed to collapse spherically. However, the precision of the focusing system and the effect of gravity, most often neglected, break the spherical symmetry of the collapse. In the experiment presented here, we generated the "most spherical" cavitation bubbles in a series of experiments by (1) running the experiments in microgravity (during the 52nd ESA Parabolic Flight Campaign), and (2) using an optimized laser focusing system. The collapse and the rebound of the bubbles were then investigated to experimentally determine the fraction of potential energy of the bubble transformed into the rebound bubble and into the shock wave at the collapse. We found that the transfer of the energy into both channels can be well predicted using a single non-dimensional parameter. A theoretical model is developed and shows good agreement with the experimental results.

[1]SNSF Grant 200020-116641

2:47PM R3.00010 Drops levitated by an air cushion: asymptotic analysis and stability

, JACCO H. SNOEIJER, Physics of Fluids Group and J. M. Burgers Centre for Fluid Dynamics, University of Twente, P.O. Box 217, 7500 AE Enschede, The Netherlands, PHILLIPPE BRUNET, Laboratoire Matière et Systèmes Complexes UMR CNRS 7057, Batiment Condorcet, 10 rue Alice Domont et L'eonie Duquet 75205 Paris cedex 13, France, JENS EGGERS, Department of Mathematics, University of Bristol, University Walk, Bristol BS8 1TW, United Kingdom — Liquid drops can be kept from touching a plane solid surface by a gas stream entering from underneath, as observed for example for Leidenfrost drops. We discuss the limit of small flow rates, for which the gap between the drop and the substrate becomes very small, to obtain a full analytical description of the drop shapes and their stability. It is found that above a critical drop radius no stationary drops can exist and that unstable drops develop a gas "chimney" that breaks the drop in its middle. We point out similarities with the breakup of drops in a microfluidic T- junctions.

Tuesday, November 22, 2011 12:50PM - 3:00PM – Session R4 Drops X: Interaction with Heterogeneous Surfaces 307

12:50PM R4.00001 Lattice Boltzmann Simulations of Drop Impact on Heterogeneous Surfaces

, KEVIN CONNINGTON, The Levich Institute, The City College of New York, TAEHUN LEE, Mechanical Engineering, The City College of New York, JEFF MORRIS, JOEL KOPLIK, The Levich Institute, The City College of New York — The dynamics of drop impact on heterogeneous substrates is important to understand from a materials research-application perspective. The phenomenon occurs in many situations, from ink-jet printing to fuel injection processes. Since drop impact is a physically complicated process, difficult to understand through experiments and theory alone, it is important to develop methods to study such a process numerically. The impact process poses many computational difficulties as well. Some of these difficulties include the tracking of a liquid-gas interface that undergoes extreme deformation in a short time period, accurately including the effects of surface tension, and realistically resolving the dynamic liquid-gas-solid contact line. Due to its kinetic nature, the Lattice Boltzmann Method(LBM) can incorporate mesoscopic physics into its formulation to resolve these difficulties, which otherwise pose significant problems for traditional continuum solvers. In our simulations, the interface is captured, instead of tracked, by keeping account of an order parameter in the fluid as in the phase field method. Here we discuss our numerical method and report results describing impact on a superhydrophobic surface for drops of large density ratio.

1:03PM R4.00002 Droplet Dynamics on Heterogeneous Substrates , DANIEL HERDE, Max Planck Institute for Dynamics and Self-Organisation, UWE THIELE, Loughborough University, STEPHAN HERMINGHAUS, MARTIN BRINKMANN, Max Planck Institute for Dynamics and Self-Organisation — Describing the dynamics of contact lines on heterogeneous substrates is important in fields ranging from coating technologies over microfluidics to enhanced oil recovery. The vast majority of studies on contact line dynamics considers the case of ideal, homogeneous substrates. Modelling chemically heterogeneous surfaces by introducing periodic variations in the wettability, we investigate the depinning and subsequent motion of 2D droplets driven by a body force. To this end, we study the time evolution of a sharp interface using Stokes dynamics together with Navier slip condition at the substrate, employing a boundary element formalism. The heterogeneity is represented by a position dependent microscopic contact angle. This allows us to quantify the modified droplet mobility, the spectrum of pinned and depinned solutions, and their dependence on periodicity and strength of the heterogeneity. Understanding the emergence of dynamic contact angles on heterogeneous substrates may open a new perspective on the relation of molecular and hydrodynamic models of contact line motion.

1:16PM R4.00003 Modelling of contact angle hysteresis on rough, non-uniform and superhydrophobic surfaces with lattice Boltzmann method[1] , K.J. KUBIAK, M.C.T. WILSON, University of Leeds, J.R. CASTREJÓN-PITA, I.M. HUTCHINGS, University of Cambridge — Contact Angle Hysteresis (CAH) is usually attributed to surface heterogeneity, contact line pinning, adsorption or interdiffusion. A model of CAH developed recently by Kubiak & Wilson is demonstrated using the lattice Boltzmann method. The model is based on the dynamic surface heterogeneity, reorientation of surface molecules under wetting liquid, physical roughness, chemical heterogeneity and liquid adhesion and evaporation. Once the surface is wetted, the local static contact angle (CA) changes from its advancing value to match the receding static CA over time Ta. When the contact line retracts, the surface recovers its initial properties corresponding to the advancing static CA over time period Te, which corresponds to the physical evaporation. Further development of the model to include surface roughness and chemical heterogeneity is presented. The model shows good agreement with experimental results for several practical configurations i.e. droplet impact and coalescence, drops on tilted surface, and drops on superhydrophobic and non-uniform surfaces etc. The extended model exhibits great potential for predictive modelling using the lattice Boltzmann method, but can be also implemented in other schemes.

[1]Research supported by EPSRC EP/F065019/1 and EP/H018913/1.

1:29PM R4.00004 A computational study of the impact of molten drops onto textured surfaces , MEHDI RAESSI, University of Massachusetts-Dartmouth, RAJKAMAL SENDHA, Indian Institute of Technology Bhubaneswar — We used an in-house, three-dimensional computational tool to study the impact and spreading of molten drops onto substrates with various surface patterns. The computational tool is GPU-accelerated and solves the mass, momentum and energy equations in the liquid phase and the conduction equation in the substrate. The drop, 40 μm in diameter, is molten alumina, which is widely used in the thermal spray coatings. The surface patterns are created by positioning cuboid obstacles that their side dimension is at least 3 μm. We investigated the effects of obstacle height, aspect ratio and spacing as well the impact velocity on the spreading dynamics and the final geometry of the drop. In our study, the impact velocity was varied from 40 to 90 m/s, the obstacle height from 1 to 5 μm, and the obstacle spacing from 2 to 26 μm. The results show that the flattened drop has a disk-shape geometry when the obstacle spacing is either smallest or largest, and that significant deformations and fingering occur at the intermediate spacings. A quantitative relation was developed between the obstacle spacing and the final spread diameter of the drop. The results show the collapse of the final spread diameter normalized by the obstacle spacing when plotted against the spacing at different impact velocity and obstacle height.

1:42PM R4.00005 Two dimensional droplet spreading over chemically heterogeneous substrates , RAJAGOPAL VELLINGIRI, NIKOS SAVVA, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK — We investigate the spreading dynamics of a partially wetting two-dimensional droplet over chemically heterogeneous substrates. The contact line singularity is removed by assuming slip at the liquid-solid interface. Assuming small contact angles and strong surface tension effects, a long-wave expansion of the Stokes equations yields a single evolution equation for the droplet thickness. The chemical nature of the substrate is incorporated through local variations in the microscopic contact angle, which appear as boundary conditions in the governing equation. By asymptotically matching the flow in the bulk of the droplet with the flow in the vicinity of the contact lines, we obtain a set of coupled ordinary differential equations for the locations of the two droplet fronts. We verify the validity of our matching procedure by comparing the solutions of the ordinary differential equations with solutions of the full governing equation. A number of interesting features that are not present in chemically homogeneous substrates are found, such as the existence of multiple equilibria, the pinning of the droplet fronts at localized chemical features and the possibility for the droplet fronts to exhibit a stick-slip-like behavior.

1:55PM R4.00006 Three-dimensional contact line dynamics on heterogeneous substrates , NIKOS SAVVA, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK — Three-dimensional contact line dynamics on heterogeneous substrates is examined theoretically by using the motion of a viscous, partially wetting droplet over a horizontal and chemically heterogeneous substrate as a model system. We utilize a long-wave model for the evolution of the droplet thickness, whereby inertial and gravitational effects are neglected and the contact angles are assumed to be everywhere small. Noteworthy is that the present formalism does not depend on an imposed, Cox-Voinov-type relation between the apparent contact angle and the contact line speed. Instead, the contact line speed is found as part of the solution, facilitated by mapping the free-boundary problem to a fixed, circular domain. Analytical progress can be made by considering perturbations from a circular contact line and asymptotically matching the flow in the two-dimensional boundary layer around the contact line with the flow in the bulk of the droplet. This matching procedure eventually leads to a set of differential equations for the Fourier coefficients of the contact line. The derived equations are solved for a few representative substrate configurations.

2:08PM R4.00007 Destiny of a drop on a fiber: from barrel to clamshell and back[1] , BURAK ERAL, J. DE RUITER, R. DE RUITER, J.M. OH, Physics of Complex Fluids Group, University of Twente, C. SEMPREBON, M. BRINKMAN, F. MUGELE, Dynamics of Complex Fluids, Max-Planck-Institute, Gottingen — Drops on cylindrical fibers are a familiar sight, for instance in the form of dew drops on spider webs. They can exist in two competing morphologies, a cylindrically symmetric barrel state completely engulfing the fiber and an asymmetric clamshell state, in which the drop sits on the side of the fiber. Despite their omnipresence and their practical relevance the physical mechanisms governing the stability of the two morphologies remained elusive. Using electrowetting-functionalized fibers we determined of the stability limits of both morphologies as a function of the two relevant control parameters, the contact angle and the liquid volume. While clamshells are found to prevail for large contact angles and small volumes, and barrels prevail for small angles and large volumes, there is also a wide range of intermediate parameter values, for which both morphologies are mechanically stable. Mapping out the energy landscape of the system by numerical minimization of the free energy we find that the barrel state is easily deformed by non-axisymmetric perturbations. From a general perspective, the demonstration of electrowetting-based reversible switching of liquid morphologies on fibers opens up opportunities for designing functional textiles and porous materials.

[1]We thank the Chemical Sciences division of the Netherlands Organization for Scientific Research (NWO-CW) for financial support.

2:21PM R4.00008 Modes of Contact Line motion on topographic substrates , CIRO SEMPREBON, STEPHAN HERMINGHAUSS, MARTIN BRINKMANN, Max Plank Institute for Dynamic and Self Organization — The link between the geometry of a regular topographic pattern, the material Contact Angle and the macroscopic Contact Angle Hysteresis is still poorly understood. Employing the apparent contact angle as a control parameter, we numerically determine the shape of a liquid interface on patterns consisting of square arrays of pillars with different cross section. Both hydrophilic and hydrophobic substrates are considered. The increase of the apparent contact angle leads to a sequence stable morphologies which may belong to different interface topologies. Eventually, the interface becomes unstable, defining the Advancing Angle. Abrupt changes of the Advancing Angle, while varying the spacing, aspect ratio and material Contact Angle of the pillars, correspond to transitions to different Advancing Modes. With the same procedure we calculate Receding Angles, and consequently the Contact Angle Hysteresis. Finally, we discuss similarities and differences among pillars with either circular and square cross sections.

2:34PM R4.00009 Investigation of Contact Angle Behavior and Stability of Drops to Airflow Forcing on Rough Surfaces , JASON SCHMUCKER, EDWARD WHITE, Texas A&M University — A method for measuring full-field, instantaneous drop interface profiles on rough surfaces has been implemented to study contact angles and stability to wind forcing on metallic surfaces with micron-scale roughness. Wind tunnel experiments are conducted to produce criteria for runback of drops and set these thresholds for measured water drops spanning a range of Bond numbers from $Bo = 0.5$ to 5 on roughness in the range of $R_A = 0.8$ to 4.9 with drop based Reynolds numbers spanning an order of magnitude. More importantly, these stability limits are tested with particular care taken to observe their relation to the behavior of both the contact line and contact angle distribution as the drop adjusts its configuration to find a stable condition until it is no longer able to do so and is blown downstream. Results such as critical shear rates and contact angles are discussed and compared with previous numerical studies in the literature such as Dimitrakopoulos [J.Fluid.Mech. 580, 2007] and Ding and Spelt [J.Coll.Sci. 599, 2008] along with experimental results such as Milne [Langmuir 25:24, 2009].

2:47PM R4.00010 Interplay of surface roughness and air pressure in splash suppression , ANDRZEJ LATKA, SIDNEY NAGEL, University of Chicago — Surface roughness has generally been found to increase the splashing of drops impacting on a solid surface [1]. However, we have recently found [2] that the effect of roughness is considerably more complex: it does promote the prompt splashing associated with rough surfaces, but also inhibits the splashing that occurs through the formation of a thin sheet. Consequently, as roughness is varied, the drop can splash through one or both of these mechanisms. Surprisingly, for sufficiently high viscosities there is a region where increasing roughness eliminates splashing entirely. We show that this result is robust and holds for a variety of liquids impinging on surfaces of different materials and geometries. The mechanism through which roughness causes splashing is intimately connected with the ambient gas. We find, as in the case of thin-sheet splashing [3], that the removal of the surrounding air suppresses prompt splashing. As the gas pressure decreases, fewer droplets are ejected until the drop completely ceases to splash. This threshold pressure increases with liquid viscosity and decreases with surface roughness and impact velocity. [1] K. Range, F. Feuillebois, *J. Colloid Interface Sci.* **203**, 16 (1998); [2] A. Strandburg-Peshkin, M. M. Driscoll, S. R. Nagel, *BAPS.* DFD.AH.7 (2009). [3] L. Xu, W. W. Zhang, S. R. Nagel, *Phys. Rev. Lett.* **94**, 184505 (2005); L. Xu, *Phys. Rev. E* **75**, 056316 (2007); M. M. Driscoll, C. S. Stevens, S. R. Nagel, *Phys. Rev. E* **82**, 036302 (2010).

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R5 CFD VII: Immersed Boundary Methods 308

12:50PM R5.00001 An implicit ghost-cell immersed boundary method with mass source and sink for simulations of moving-body problems with control of spurious pressure oscillations[1] , JINMO LEE, DONGHYUN YOU, Department of Mechanical Engineering, Carnegie Mellon University — For moving body problems where the conventional immersed-boundary methods are employed, the simulation results are often contaminated by spurious pressure oscillations. The spurious pressure oscillations are known to be produced by the violation of mass conservation across interfaces between fluid and moving bodies and to be a function of mainly the grid spacing and time-step size. In the present work, we develop a new immersed boundary method which can control and significantly reduce the spurious pressure oscillations. A ghost-cell immersed boundary method is coupled with a mass source and sink algorithm to better conserve mass around boundary interfaces. A fully-implicit time integration scheme is employed for enhanced control of the time- step size thereby allowing better control of the spurious force oscillations on given mesh resolution. A novel method for treating multiple fresh and dead cells due to the use of a large time-step size is also developed. The present fully-implicit ghost-cell immersed boundary method coupled with a mass source/sink algorithm is demonstrated to significantly reduce spurious pressure oscillations thereby providing accurate and stable solutions for moving body problems.

[1]Supported by the Office of Naval Research Grant N000141110652.

1:03PM R5.00002 A volume penalization method for incompressible flows and scalar advection-diusion with moving obstacles , KAI SCHNEIDER, BENJAMIN KADOCH, DMITRY KOLOMENSKIY, M2P2-CNRS & CMI Aix-Marseille University, France, PHILIPPE ANGOT, LATP-CNRS & CMI Aix-Marseille University, France — A volume penalization method for imposing homogeneous Neumann boundary conditions in advection-diffusion equations is presented. Thus complex geometries which even may vary in time can be treated efficiently using discretizations on a Cartesian grid. A mathematical analysis of the method is conducted first for the one-dimensional heat equation which yields estimates of the penalization error. The results are then confirmed numerically in one and two space dimensions. Simulations of two-dimensional incompressible flows with passive scalars using a classical Fourier pseudo-spectral method validate the approach for moving obstacles. The potential of the method for real world applications is illustrated by simulating a simplified dynamical mixer where for the fluid flow and the scalar transport no-slip and no-flux boundary conditions are imposed, respectively.

1:16PM R5.00003 A hybrid Pseudo-spectral Immersed-Boundary Method for Applications to Aquatic Locomotion , ZHENG REN, DAVID HALL, KAMRAN MOHSENI — A hybrid pseudo-spectral immersed boundary method is developed for application in marine locomotion. Spatial derivatives are calculated using pseudo-spectral method while a 2nd-order Runge-Kutta scheme is used for time integration. The singular force applied on the immersed boundary is obtained using a direct forcing method. To avoid Gibb's phenomenon in the spectral method, we regularize the force by smoothing it over several grid cells. This method has the advantage of spectral accuracy and the flexibility to model irregular, moving boundaries on a Cartesian coordinate without complex mesh generation. The method is applied to examine locomotion of jellyfish for both jetting and paddling jellyfish.

1:29PM R5.00004 Wall-models for immersed-boundary methods[1] , ANTONIO POSA, Politecnico di Bari, ELIAS BALARAS, The George Washington University — Immersed boundary (IB) methods, where the requirement for the computational grid to conform to solid boundaries is relaxed, have been widely used in a variety of applications. In the majority of cases IB methods are usually coupled to Cartesian solvers, and tackle low to moderate Reynolds number problems. Applications at high Reynolds numbers are prohibitively expensive since an increase in the wall-normal resolution can only be achieved by refining the computational grid in all coordinate directions. A solution to this problem is the development of wall-models for IB methods that can compensate for the luck of resolution in both laminar and turbulent regimes. In the present study we propose novel wall treatment based on a two-layer zonal approach, where the solution of a simplified set of equations is solved near the wall for the purpose of providing an accurate estimate of the near wall flow using information form the coarse underlying Navier-Stokes grid. Various models based on different levels approximation will be presented. Results will be shown for canonical cases such as the flow around a smooth cylinder and around a sphere at various Reynolds numbers.

[1]Supported by ONR

1:42PM R5.00005 Fluid-Structure Interaction for Flapping Flexible Wings with Large Mass Ratio[1] , MIN XU, MINGJUN WEI, New Mexico State University — A strong-coupling approach has been successfully used in our previous study for the fluid-structure interaction of flapping flexible wings. However, when the mass ratio of wing and fluid is considered, we are facing a problem to solve Poisson equation with discontinuous coefficients. As the mass ratio increases, normal algorithm for solving the above equation becomes costly and unstable. In this work, we applied the Black Box Multigrid Conjugate Gradient Preconditioned Method (Box-MGPCG) and a smoothing function to overcome the problem. The new algorithm shows consistent efficiency for mass ratio up to 1000. Therefore, it allows us to study the effect of large mass ratio to the performance of flapping flexible wings. Simulation results are also presented here.

[1]Supported by ARL through Army High Performance Computing Research Center

1:55PM R5.00006 Simulation of flow over a sphere in a boundary layer using a GPU accelerated IB-LBM[1] , WONHO BAE, JUNG-IL CHOI, Dept. CSE, Yonsei University — Numerical simulations of flow over a sphere embedded in a laminar boundary layer are conducted for characterizing the effects of wall proximity on the drag and lift forces acting on the sphere. The wall proximity is defined as the distance from the wall to the center of sphere. We utilize an immersed boundary-lattice Boltzmann method (IB-LBM) with a multi-direct forcing technique (Suzuki & Inamuro, Computers & Fluids 2011) and combine the present method with a multi-block method (Yu et al., IJNMF 2002) for refining lattices near the sphere. We implement the present IB-LBM into a Graphical Processing Unit (GPU) using a PGI CUDA Fortran programming environment for accelerating the computations. We perform benchmark tests based on simulations of flow over a sphere in a free-stream for validations of the present IB-LBM and evaluations of the performance of the GPU implementation. The results of the drag and lift forces on the sphere according to the wall proximity will be shown in the final presentation.

[1]Supported by World Class University (WCU) program of NRF (R31-10049).

2:08PM R5.00007 Gust effects on a freely falling plate[1] , HUI WAN, HAIBO DONG, ZONGXIAN LIANG, Wright State University, FSRG TEAM — Depending on the Reynolds number and the Froude number, a freely falling plate usually performs one of the following four types of motion, flutter, tumble, steady or chaos fall. It is interesting to know that if and how a gust changes the falling status of a plate. In this work, Direct Numerical Simulations (DNS) will be conducted to study the effects of gust on the freely falling plate by varying the gust amplitude, frequency, and phase relative to the falling plate. Especially, for a plate lies in the chaotic (transitional) region, how its motion be affected as a response to the gust will be discussed.

[1]NSF CBET-1055949

2:21PM R5.00008 Three-dimensional simulations of the compressible, low-Reynolds number, flow around a hohlraum in a fusion chamber[1] , HASIB UDDIN, CARLOS PANTANO, University of Illinois at Urbana-Champaign, GWEN LOOSMORE, Lawrence Livermore National Laboratory — Three-dimensional immersed interface simulations of the flow around a hohlraum in a fusion chamber will be presented. The high injection velocity of the hohlraum results in important compressibility effects while the Reynolds number is quite low due to the high temperature in the chamber (up to 8000K); still well within the continuum limit. A stable recirculation region forms behind the object. In addition, the hohlraum surface temperature is very low to preserve the hydrogen pellet at cryogenic conditions. These conditions are not commonly encountered in flows around blunt objects since high Mach number usually implies high Reynolds number. We study the effects of spin rate at different angles of attack and free flight dynamics. The formation of near surface secondary flow patterns is discussed as well as the distribution of the heat flux.

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

2:34PM R5.00009 A Hybrid Immersed Boundary-Immersed Interface Method for Cell Tracking in Microdevices , MOHAMMAD HOSSAN, PRASHANTA DUTTA, ROBERT DILLON, Washington State University — The manipulation of cells in microfluidic devices has become routine for biomedical applications such as cell sorting and trapping. To date most of the designs used for cell manipulation are based on experimental trial and error. A fast and accurate numerical algorithm can provide important insight into the design of these devices. In this study, a hybrid immersed boundary-immersed interface method is developed to study the complex behavior of cells in liquid. The immersed boundary method provides an accurate prediction of particle motion in a fluid while the immersed interface method gives second-order accurate solutions for the ion concentrations and electrostatic potential in the presence of moving cells. Both methods employ a fixed computational grid without the need for remeshing at each time step. Cells of different size, shape and charge are allowed to move under both hydrodynamic and electrokinetic forces. Moreover different channel geometries are considered to obtain the best trapping and separation performance. The present immersed boundary-immersed interface model is particularly suitable for bioMEMS devices as this method can accurately predict viscous and electrostatic forces as well as particle velocity, location, and particle membrane deflection.

2:47PM R5.00010 Large Eddy Simulation of Motion-Induced Contaminant Transports in Room Compartments[1] , JUNG-IL CHOI, Dept. CSE, Yonsei University, JACK EDWARDS, Dept. MAE, North Carolina State Univ. — Large eddy simulation (LES) of contaminant transports due to complex human and door motions is conducted for characterizing the effect of the motion-induced wakes on the contaminant transports in room compartments where a contaminated and clean room are connected by a vestibule. We utilize a LES technique with an immersed-boundary method for moving objects (Choi et al., JCP 2007; Choi and Edwards, Indoor Air 2008) and extend the technique to include Eulerian descriptions of gas-phase contaminant transport as well as thermal energy transfer. We demonstrate details of contaminant transport due to human- and door-motion induced wake development during a short-duration event involving the movement of a person (or persons) from a contaminated room, through a vestibule, into a clean room. Parametric studies that capture the effects of human walking pattern, door operation, over-pressure level, and vestibule size are systematically conducted. The results of parameteric studies will be shown in the final presentation.

[1]Supported by DARPA/SPO program (HR0011-05-C-0157) and WCU program (R31-10049) of NRF.

Tuesday, November 22, 2011 12:50PM - 2:47PM –
Session R6 Focus Session: Evaporative Self-Assembly of Micro-and Nano-Particles 309

12:50PM R6.00001 Drying of complex fluids at a moving contact line[1] , CHING HSUEH, FREDERIC DOUMENC, BEATRICE GUERRIER — We study the pattern formation induced by drying colloidal suspensions in a vertical Hele-Shaw cell immersed in a reservoir [1]. The contact line velocity can be well controlled by pumping out the solution from the reservoir. At low capillary number, we observe stick-slip motion and periodic strip deposition [2]. We measured the pinning force variation and the wavelength. We systematically vary the following parameters: receding velocity, evaporation rate, concentration, particle size, and pH of the suspension. Results allow determining the power law governing the pinning force variation. The pH, which has no effect on the pinning force variation, changes the deposition morphology significantly. Finally, we present a detailed comparison between colloidal suspensions and polymer solutions.

[1] H. Bodiguel, F. Doumenc, B. Guerrier EPJ-ST 166, 29-32 (2009)
[2] H. Bodiguel, F. Doumenc, B. Guerrier Langmuir, 13, 10758-10763 (2010)

[1]This work was supported by the Marie Curie Initial Training Network: MULTIFLOW.

1:03PM R6.00002 Dynamics of freely moving plates connected by a shallow liquid bridge[1] , AMIR GAT, Caltech, HOMAYUN NAVAZ, Kettering University, MORTEZA GHARIB, Caltech — We study the dynamics of freely moving solid bodies connected by a shallow liquid bridge via analytic and experimental methods. The gap between the solid bodies is used as a small parameter within a lubrication approximation, reducing the problem to an Abel equation of the second kind. Analysis of the governing differential equation yields two novel physical phenomena: (1) An impulse-like peak in the force applied by the liquid bridge on the solid bodies, obtained from a uniform asymptotic solution for small Capillary numbers. (2) Both linear and non-linear oscillations of the system for the case of surfaces with low wettability, obtained from small perturbations of the system around the equilibrium point. An experimental setup examining the motion of freely moving solid bodies was constructed, yielding experimental data which compared favorably with the analytic results and specifically displayed the predicted oscillations and impulse-like peak of the applied force. The application of the current analysis to the micro-manipulation of solid bodies and possible future research directions are discussed.

[1]This project was supported by the Defense Threat Reduction Agency (DTRA), Award Number: 330233-A.

1:16PM R6.00003 Real Time Electron Microscope Imaging of Nanoparticle Motion Induced by a Moving Contact Line[1] , JOSEPH GROGAN, HAIM BAU, University of Pennsylvania — With the high resolution of the electron microscope, we imaged the interactions between receding and advancing contact lines and surface-bound nanoparticles. The experiments were carried out with a custom-made liquid cell, dubbed the nanoaquarium. The nanoaquarium seals a thin liquid layer between two thin, electron-transparent membranes, and allows one to image processes in liquid media with electrons. We observe that the nanoparticles are ejected in the wake of the receding contact line, and pushed by the advancing contact line. A simple mathematical model that accounts for surface tension and disjoining pressure effects is constructed to interpret this curious phenomenon.

[1]The work was supported, in part, by the NSF grant 1066573.

1:29PM R6.00004 Field-directed assembly of nanoparticles[1] , ERIC FURST, University of Delaware — The use of external fields to direct the assembly of colloidal suspensions, combined with new particle shape symmetries that couple strongly to such fields, is a powerful means for creating and tailoring materials with unique mechanical, optical and electronic properties [1]. I will present the evaporative assembly of nanostructured thin films from ellipsoidal titania nanoparticles. The deposition process is directed by an electric field. As the evaporation front recedes, a uniform film with thicknesses of 1-10 μm is deposited on the substrate. The films exhibit a large birefringence and high packing fraction due to the orientation of the particles. When the frequency is lowered, the particle orientation undergoes a parallel-random-perpendicular transition with respect to the field direction. The orientation dependence on field frequency and strength is explained by the polarizability of ellipsoidal particles. Particle orientation in the films also leads to anisotropic mechanical properties, which are manifested in their cracking patterns. In all, field-directed assembly of anisotropic particles provides a powerful means for tailoring nanoparticle film properties *in situ* during the deposition process.

[1] Grzelczak et al. Directed Self-Assembly of Nanoparticles. ACS Nano 4, 3591-3605 (2010).

[1]Funding from the Department of Energy Basic Energy Sciences (DE-FG02-09ER46626) is gratefully acknowledged.

1:42PM R6.00005 Suppressing the coffee stain effect: how to control colloidal self-assembly in evaporating drops using electrowetting[1] , BURAK ERAL, DILEEP MAMPALLIL AUGUSTINE, MICHEL DUITS, FRIEDER MUGELE, Physics of Complex Fluids Group, University of Twente, PHYSICS OF COMPLEX FLUIDS GROUP, UNIVERSITY OF TWENTE TEAM — We study the influence of electrowetting on the evaporative self-assembly and formation of undesired solute residues, so-called coffee stains, during the evaporation of a drop containing non-volatile solvents. Electrowetting is found to suppress coffee stains of both colloidal particles of various sizes and DNA solutions at alternating (AC) frequencies ranging from a few Hertz to a few tens of kHz. Two main effects are shown to contribute to the suppression: (i) the time-dependent electrostatic force prevents pinning of the three phase contact line and (ii) internal flow fields generated by AC electrowetting counteract the evaporation driven flux and thereby prevent the accumulation of solutes along the contact line Please see the link below for a short presentation and movies: http://www.youtube.com/watch?v=xwipCVZnN4E

[1]We thank the Chemical Sciences division of the Netherlands Organization for Scientific Research (NWO-CW) for financial support (ECHO grant).

1:55PM R6.00006 Vibration Assisted Convective Deposition , TANYAKORN MUANGNAPOH, ALEXANDER WELDON, PISIST KUMNORKAEW, JAMES GILCHRIST, Center for Advanced Materials and Nanotechnology, Department of Chemical Engineering, Lehigh University, Bethlehem, Pennsylvania 18015 — A novel strategy for improving a convective deposition of aqueous binary suspensions of colloidal microspheres and nanoparticles was experimentally examined and reported. By adding a substrate vibration, an enhancement in deposited film qualities was observed. Moreover, by using this technique, it was easier and faster to obtain monolayer structures. In this experiment, we varied the amplitudes of substrate vibration between 0-330 μm. The quality of thin films was characterized by using a confocal laser scanning microscope and an image analysis. The motion of an interfacial liquid surface and the change in an evaporate rate due to a substrate vibration played an important role in an improvement of the deposition process. The monolayer structures formed from this rapid process can be used in a variety of optical, chemical, and biochemical sensing applications such as a LEDs device, a membrane separation and a cell capturing.

2:08PM R6.00007 Interaction of bi-dispersed particles with contact line in inkjet-printed evaporating colloidal drops , VIRAL CHHASATIA, YING SUN, Drexel University — The deposition behavior of inkjet-printed aqueous colloidal mixture of micro and nanoparticles onto a glass substrate with systematically varied wettability has been investigated using fluorescence microscopy. Real-time bottom-view images show that particles inside an evaporating drop rearrange themselves near the drop contact line according to their sizes, where smaller particles tend to deposit closer to the contact line compared to the larger ones. By increasing substrate wettability, particles in the bi-dispersed mixture can be further separated compared to those on substrates of poor wettability. This is primarily because during different stages of evaporation, the interplay of surface tension, drag due to evaporative flow, and particle-substrate interactions, rearrange particles inside a colloidal drop near the contact line region. Forces acting on particles determine the extent to which particles enhance contact line pinning, which ultimately determines the final deposition morphology of particles from a bi-dispersed colloidal mixture. The effects of particle size contrast, particle volume fraction, and substrate surface energy on particle separation are examined in detail.

2:21PM R6.00008 Fast Evaporation of Spreading Droplets of Colloidal Suspensions , KARA MAKI, Rochester Institute of Technology, SATISH KUMAR, University of Minnesota — When a coffee droplet dries on a countertop, a dark ring of coffee solute is left behind, a phenomenon often referred to as "the coffee-ring effect." A closely related yet less-well-explored phenomenon is the formation of a layer of particles, or skin, at the surface of the droplet. In this work, we explore the behavior of a mathematical model that can qualitatively describe both phenomena. We consider a thin axisymmetric droplet of a colloidal suspension on a horizontal substrate undergoing spreading and rapid evaporation. The lubrication approximation is applied to simplify the mass and momentum conservation equations, and the colloidal particles are allowed to influence droplet rheology through their effect on the viscosity. By describing the transport of the colloidal particles with the full convection-diffusion equation, we are able to capture depthwise gradients in particle concentration and thus describe skin formation, a feature neglected in prior models of droplet evaporation. Whereas capillarity creates a flow that drives particles to the contact line to produce a coffee-ring, Marangoni flows can compete with this and promote skin formation. Increases in viscosity due to particle concentration slow down droplet dynamics, and can lead to a significant reduction in the spreading rate.

2:34PM R6.00009 Capillary-Driven Convective Assembly of Colloidal Monolayers , ALEXANDER WELDON, PISIST KUMNORKAEW, TANYAKORN MUANGNAPOH, JAMES GILCHRIST, Lehigh University — Convective self-assembly is a powerful method for the deposition of particle thin films. We investigate the coupling between suspension properties and the deposition process during convective deposition of unary colloidal silica microspheres as well as the use of nanoparticles as packing aids. We can tune suspension and deposition properties to deposit submonolayer, monolayer, or multilayer morphologies. Thin films are analyzed via high speed confocal and scanning electron microscopy in order to generate local dynamic data describing the deposition process as well as the long-range structure of deposited thin films.

Tuesday, November 22, 2011 12:50PM - 3:00PM – Session R7 Turbulent Boundary Layers VIII: Geophysical 310

12:50PM R7.00001 Disruption of bottom log-layer in LES of Langmuir circulation in shallow seas , NITYANAND SINHA, ANDRES E. TEJADA-MARTINEZ, University of South Florida, CHESTER E. GROSCH , GUILLAUME MARTINAT, Old Dominion University — We report on disruption of the log-layer in the resolved bottom boundary layer in large- eddy simulations (LES) of full-depth Langmuir circulation (LC) in a wind-driven shear current in neutrally-stratified shallow water. LC consists of parallel counter rotating vortices that are aligned roughly in the direction of the wind and are generated by the interaction of the wind-driven shear with the Stokes drift velocity induced by surface gravity waves. The disruption is analyzed in terms of mean velocity, budgets of turbulent kinetic energy (TKE) and budgets of TKE components. For example, in terms of mean velocity, the mixing due to LC induces a large wake region eroding the classical log- law profile within the range $90 < z+ < 200$. The dependence of this disruption on wind and wave forcing conditions is investigated. Results indicate that the amount of disruption is primarily determined by the wavelength of the surface waves generating LC. These results have important implications on turbulence parameterizations for Reynolds-averaged Navier-Stokes simulations (RANSS) of the coastal ocean. Preliminary simulations highlight the need for turbulence models taking into account log-layer disruption by LC, as RANSS with the k-epsilon model is unable to capture this disruption.

1:03PM R7.00002 ABSTRACT WITHDRAWN –

1:16PM R7.00003 Dispersion length scales within the urban canopy[1] , PABLO HUQ, University of Delaware, PASQUALE FRANZESE, George Mason University — We discuss the results of lab experiments on three model urban canopies with small, medium and large building aspect ratios to examine the physics of dispersion within the urban canopy from a near-ground continuous point source of passive scalar. The model urban canopies had aspect ratios of building height to width (H/w) = 0.25, 1, 3. Measurements were taken of the turbulent velocity and scalar fields. Plume spreads, concentrations and distance from the source were non-dimensionalized using urban canopy length, time and velocity scales based on the geometry of the buildings. The scaling collapses the data for all three aspect ratios. A model to describe the results is developed. The model is based on a simple Gaussian formulation where the diffusion coefficients are determined by the theories of Taylor (1921) in the horizontal plane, and Hunt and Weber (1979) to account for the vertically inhomogeneous turbulence.

[1]NSF Grant AGS 0849190 and 0849191

1:29PM R7.00004 Scale separation effects in turbulent boundary layers[1]

CALEB MORRILL-WINTER, University of New Hampshire, JOSEPH KLEWICKI, University of New Hampshire, University of Melbourne — The velocity and vorticity field interactions that underlie the mean mechanism of turbulent inertia and the wall-normal variation of turbulence kinetic energy are investigated over a wide variation in Reynolds number. Existing well-resolved laboratory data, $\delta^+ = 375$, 970 & 1500, and data from the atmospheric surface layer over Utah's west desert, $\delta^+ = 890,000$, are used to establish the relevant statistical and spectral properties. The influences of scale- separation, as well as the scale selection phenomena first observed by Priyadarshana et al. (*J. Fluid Mech.* **570**, 2007) are of interest. Scale-separation is quantified using both the difference and ratio of the peak frequencies of the relevant velocity and vorticity components. The scale selection phenomena is clarified by examining the relative behavior of the spectra and associated co-spectra as a function of both y^+ and δ^+. Near the wall, scale- separation is due to the rapid spatial confinement of the vortical motions, while the scale selection correlates with the vorticity spectra. Away from the wall, scale-separation is due to the spatial dispersion of the vortical motions, and the scale selection correlates with the velocity spectra.

[1]The support of the ONR (N000140810836, grant monitor Ronald Joslin) is gratefully acknowledged.

1:42PM R7.00005 On the distribution of the streamwise fluctuation velocity in wall bounded flows

P. HENRIK ALFREDSSON, RAMIS OERLUE, ANTONIO SEGALINI, Linne FLOW Centre, KTH Mechanics, Royal Institute of Technology, Stockholm, Sweden — The streamwise velocity fluctuations in wall-bounded flows has recently received renewed interest. Measurements and simulations at low Reynolds numbers (Re), as well as high Re wind tunnel studies and data from the atmospheric boundary layer (ABL) are at hand but show sometime conflicting trends with Re. However, high Re data often have some uncertainties associated with them, such as poor spatial resolution for laboratory data or other uncertainties associated with ABL experiments. Several models for the wall normal distribution of u_{rms} have recently been proposed, based on various physical ideas together with empirical inputs. Here we propose a new model based on two observations: a) u_{rms} normalized with the local velocity U decreases linearly with respect to U/U_∞ in the outer part of the flow, b) in the inner region u_{rms}/U deviates from the linear trend at a specific value of $U/u_\tau (\approx 17)$, where u_τ is the friction velocity. Using this information it is possible to formulate a composite description of u_{rms} in the wall normal direction for all Reynolds numbers. This shows two important results, namely the increase in u_{rms}/u_τ with Re as well as a prediction of a second "outer" maximum when Re is high enough, a debated feature that has been observed in ABL experiments as well as in some laboratory experiments at high Re.

1:55PM R7.00006 The effect of atmospheric stability on the energetic contribution of the large scale structures in turbulent boundary layers

MICHELE GUALA, LEONARDO P. CHAMORRO, SAFL, Dep. Civil Engineering, University of Minnesota, MN — Turbulent boundary layer measurements in wind tunnels and in the near neutral atmospheric surface layer outlined a significant contribution of the large scale motions to turbulent kinetic energy and Reynolds stresses for a wide range of Reynolds number, providing evidence of complex scale interactions across the wall region. In order to understand the effect of the large scales on the near wall turbulence and extend the predictive models of amplitude modulation to more realistic atmospheric conditions, different thermal stability conditions must be explored. In this study, experiments are performed in the atmospheric wind tunnel of the St. Anthony Falls Laboratory independently controlling air flow and floor temperatures. Measurements of fluctuating temperature simultaneously with the streamwise and wall normal velocity components are obtained with an ad hoc calibrated and customized triple-wire sensor. Scaling quantities and the dominant terms in the turbulent kinetic energy and temperature variance budget equations are estimated and discussed. A comparative analysis of the weakly stable, convective and neutral conditions based on the power spectra of the streamwise, wall normal and Reynolds stress contributions is presented. Appreciable differences in the energetic contributions of the large scales were observed.

2:08PM R7.00007 Cyclic instabilities, turbulence and heat transfer in rotating channel flow simulations

GEERT BRETHOUWER, PHILIPP SCHLATTER, ARNE JOHANSSON, DAN HENNINGSON, FLOW, KTH Mechanics, Sweden — Fully developed turbulent channel flow subject to spanwise rotation including a passive scalar is studied by direct numerical simulations. The Reynolds number based on the bulk velocity and channel half width is up to 30000 and the rotation rate covers a broad range. At moderate rotation rates the flow partly relaminarizes on the stable side of the channel and in some cases turbulent spots or oblique laminar and turbulent bands can be identified. The turbulent Prandtl number is close to one in non-rotating channel flow, but is it much smaller if the flow is rotating. At high rotation rates and sufficiently high Reynolds numbers cyclic instabilities are observed. The time scale of the instabilities is much longer than any turbulent time scale. Further analysis show that these instabilities are caused by exponentially growing Tollmien-Schlichting (TS) waves which can develop due to the weak turbulence in rapidly rotating channel flow. When the amplitude is large these TS waves become unstable and break down into intense turbulence which decays because of the rotation. After some time the TS waves start growing again when the turbulence is sufficiently weak and the whole process repeats itself. The cyclic instabilities lead to intense bursts of turbulence, strong wall shear stresses and high heat transfer rates.

2:21PM R7.00008 The role of stability in modulating the structure and transport efficiency of turbulence in the atmospheric surface layer

DAN LI, ELIE BOU-ZEID, Princeton University — A vast body of literature has emerged over the last few decades on the topology, dynamics, and role of coherent structures in turbulent boundary layer flows. The applicability of this knowledge to geophysical flows is problematic due to the often-dominant role of buoyancy. Here we aim to investigate the effect of buoyancy on coherent turbulent structures, with a focus on the ties between these structures and turbulent transport. The results confirm that the topology of the coherent structures is very sensitive to stability. The findings point to a gradual transformation of the structures from hairpin vortices (or horizontal rolls) to thermals, as the upward buoyancy flux increases. More importantly, this change induces a decorrelation of the momentum and scalar fluxes in the surface layer and significant change in the relative efficiencies of momentum and scalar transport. Scalars are transported much more efficiently under unstable conditions. These findings provide a better framework for including the effect of stability in turbulent transport models and open the way for more advanced models based on a better understanding of turbulent scale-interactions under different stabilities.

2:34PM R7.00009 Viscous boundary layers in turbulent Rayleigh-Bénard convection[1]

LING LI, RONALD DU PUITS, ANDRÉ THESS, Ilmenau University of Technology — Thermal convection at high Rayleigh number is a basic and important ingredient for the motion of air or the flow of water in the atmosphere and in the ocean. However, particularly in case of highly turbulent flows the knowledge about the temperature and the velocity field is still limited. Highly resolved 3d-Laser Doppler Velocimetry measurements in a large-scale Rayleigh-Bénard experiment with air at Rayleigh numbers up to 10^{10} have been carried out and presented by our group on 2010's APS conference. All three velocity components have been measured simultaneously in the vicinity of the cooling plate in the central axis of the cylindrical sample. We found that the profile of the mean wall-normal velocity tends to zero. In the contribution of this year we will enhance the understanding of the heat transport by presenting the fluctuations of the wall-normal velocity component and the temperature. Again, we estimate the profile of the local heat flux from the independently measured velocity and temperature data.

[1]We wish to acknowledge the DFG (Grants No. Pui 436/1-1) as well as the CSC (Grants No. 2009608062).

2:47PM R7.00010 Reynolds and swirl number effects on turbulent pipe flow in a 90 degree pipe bend , ATHANASIA KALPAKLI, RAMIS OERLUE, P. HENRIK ALFREDSSON, CCGEx, KTH Mechanics, Royal Institute of Technology, Stockholm, Sweden — Flows in pipe bends have been studied extensively over the last decades due to their occurrence both in the human respiratory and blood systems as well as in many technical applications. The centrifugal effect of the bend may give rise to Dean vortices and the behaviour of these has been of particular interest. While their motion has nicely been illustrated in laminar flows, the picture of their motion in turbulent flows remains rather blurred. Within the framework of the present work, fully developed turbulent pipe flow from a 100 diameter (D) long pipe is fed to a 90° bend and the flow field at $0.5D$ downstream the bend has been studied by means of Time-Resolved Stereoscopic Particle Image Velocimetry, covering a Reynolds number range from 7000 to 34000 based on bulk velocity (U_b) and D. Additionally, a well defined swirl profile could be introduced by rotating the $100D$ long straight pipe along its axis, yielding a variation in swirl number (S), defined as the ratio between the azimuthal velocity of the pipe wall and U_b, from 0 (the non-rotating case) to 1.2. The three-dimensional time-averaged and instantaneous flow field illustrating the symmetrical Dean vortices for $S = 0$ and the influence by the swirling motion for $S \neq 0$, the so-called "swirl-switching phenomenon," as well as the large-scale structures will be presented and discussed.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R8 Non-Newtonian Flows III 311

12:50PM R8.00001 The effects of long-chain polymers on tip vortex flow and cavitation inception , QUAN ZHANG, CHAO-TSUNG HSIAO, GEORGES L. CHAHINE, Dynaflow, Inc — Experiments have shown that propeller/hydrofoil tip vortex cavitation can be suppressed by properly injecting dilute polymer solutions at the tip. However, the mechanisms for this phenomenon are not well understood yet. To understand better the underlying flow physics the tip vortex flow generated by a rotating propeller in water and a dilute polymer solution (FENE-P model) was numerically simulated. It is found that the vortex flow structure is changed by the non-Newtonian features of polymers. Phenomenally the vortical rotation in a polymer solution is slower and the vortex center pressure is higher than in water. The non-Newtonian stress is much stronger than the Newtonian stresses in water. To further understand the non-Newtonian stresses contribution, the FENE-P model is also applied to a simplified quasi-cylindrical vortex. It is found analytically that in addition to the three normal stresses that are expected to be quadratic in the shear rate, one of the shear components is also quadratic. We also studied polymer effects on the dynamics of a bubble nucleus in the tip vortex. The bubble was found to grow to an elongated large cavity in water while it collapses in the polymer solution for the same cavitation number. This work was supported by the Office of Naval Research, Contract N00014-04-C-0110, monitored by Dr. Ki-Han Kim.

1:03PM R8.00002 Modeling Vortex Cavitation Inception Delay in a Swirl Chamber by Polymer Injection[1] , J. MA, Q. ZHANG, C.T. HSIAO, G.L. CHAHINE — Experimental studies have shown tip vortex cavitation can be delayed with injection of drag reducing dilute polymer solutions. We present here numerical simulations conducted to understand the mechanisms responsible for cavitation suppression with local polymer injection. A canonical flow in a linear vortex chamber was simulated by using the NS solver, 3DYNAFS-VIS©, equipped with a FENE-P viscoelastic model for the polymer solution and a transport equation to track its concentration. The simulation showed that injection of dilute polymer can delay cavitation inception at a much lower injection flow rate than needed with massive injection of water or a higher viscosity liquid. Injection of polymer increases the pressure along the vortex axis and a much earlier vortex breakdown created by the elasticity of the polymers appears to be responsible for the strong modification of the flow character. This results in a fast reduction of the rotational velocity, increase of the pressure, and delay of cavitation inception. The dependency of polymer effects on the injection flow rate and polymer concentration was also investigated, finding good consistency with experimental observations.

[1]ONR Contract N00014-04-C-0110, monitored by Dr. Ki-Han Kim.

1:16PM R8.00003 Viscoelastic instabilities in a 3D Stokes-Oldroyd-B fluid , BECCA THOMASES, UC Davis, MICHAEL SHELLEY, Courant Institute — The consequences of three-dimensional viscoelastic instabilities are examined numerically using the Oldroyd-B model in the low Reynolds number (Stokes) regime. The fluid is driven by a simple time-independent forcing that, in the absence of viscoelastic stresses, creates a four-roll mill in (x,y) which is constant in z. It is now known that such forcing will force the 2d version of this system into symmetry breaking and flow mixing. Here we find that at sufficiently large, but O(1), Weissenberg number, 3d perturbations grow exponentially and lead to complex three-dimensional flow dynamics which can differ markedly from the 2d case.

1:29PM R8.00004 ABSTRACT WITHDRAWN –

1:42PM R8.00005 Study of the behavior of rising bubbles in a Boger-type fluid , J.R. VELEZ-CORDERO, D. SAMANO, R. ZENIT, Universidad Nacional Autonoma de Mexico — Particle aggregation is a common phenomenon observed in viscoelastic multiphase flows. In this work a new effect has been observed to occur in monodispersed bubbly flows in a Boger-type fluid. It was found that the dispersion of bubble changes dramatically depending on the bubble size: if the diameter of the bubbles is small, large vertical clusters are formed; on the other hand, the bubble assembly rises in a dispersed manner if the bubble size is increased. To understand the condition for which agglomeration occurs two additional experiments were conducted: the interaction of two side-by-side bubble chains was analyzed; and, the unsteady behavior of the first normal stress difference was studied in a rheometric flow. These analyses suggest that there is a process of accumulation of elastic stress; when the accumulated elastic stress surpasses the viscous repulsive stress, aggregation can occur even at supercritical speeds. Interestingly, the two bubble diameters tested in the bubbly flow experiments are above and below the critical diameter for which the velocity of an isolated bubble becomes discontinuous, the so-called bubble velocity discontinuity. This suggests that the bubble dispersion improvement could result from the modification of the gas-liquid interface.

1:55PM R8.00006 Pertrurbation theory for dynamic behavior of a sphere settling in a viscoelastic fluid , MATTHEW NICK MOORE, Courant Institute of Mathematical Sciences, BIN LIU, Brown University School of Engineering, JUN ZHANG, New York University Department of Physics, Courant Institute, MICHAEL J. SHELLEY, Courant Institute of Mathematical Sciences, APPLIED MATHEMATICS LABORATORY, COURANT INSITUTE TEAM — We present a new perturbation theory for the motion of a rigid sphere settling in a viscoelastic Oldroyd-B fluid that can be generalized to other scenarios of viscoelastic fluid-structure interaction. In contrast to previous perturbation theories, the perturbative expansion variable is not the Weissenberg number, but instead it is a parameter measuring the feedback of the viscoelastic stress into the fluid momentum. This allows for accurate calculations at large Weissenberg numbers. Previous experiments, including our own, have documented that a sphere overshoots its terminal velocity on a transient timescale comparable to the fluid relaxation time. Our theory predicts this behavior as well as a non-trivial dependence of the drag on the Weissenberg number. I will also discuss experiments in which periodic forcing is applied to a body moving through a viscoelastic fluid, and the perturbation theory is used as a predictive tool.

2:08PM R8.00007 Phase syncronization of swimming infinite sheets in viscoelastic fluids[1] , JOHN CHRISPELL, Indiana University of Pennsylvania, MICHAEL SHELLEY, Courant Institute of Mathematical Science, LISA FAUCI, Tulane University — A Navier-Stokes/Oldroyd-B immersed boundary algorithm is used to examine the interaction of swimming infinite sheets with a viscoelastic fluid. In particular, we examine the spatial and temporal evolution of the polymer stress field. The effects of the bulk viscoelasticity on hydrodynamic synchronization of swimming sheets and sheets swimming next to solid walls is analyzed.

[1]Supported in part by NSF DMS 0652775

2:21PM R8.00008 Effects of viscoelasticity on the migration of a viscous drop in a shear flow near a wall[1] , SWARNAJAY MUKHERJEE, KAUSIK SARKAR, University of Delaware — Dynamics of a drop migrating in a shear flow of a viscoelastic liquid (FENE-CR) near a wall is numerically investigated. Viscoelasticity hinders migration, and it is explained by investigating the viscoelastic forces around the drop. The orientation angle and the lateral migration velocities both decrease linearly with increasing viscoelasticity (Deborah number and amount of polymer viscosity). The enhanced curvature of the streamlines above the drop adds to this effect, it being more prominent for smaller deformation at lower capillary numbers. The slip velocity of the drop decreases with increasing Deborah number. For viscosity matched systems, the initial position does not influence the migration for the low values of Deborah number considered. However, at higher viscosity ratios, initial position plays a role. Increasing the viscosity ratio lowers the migration velocity and addition of viscoelasticity decreases it further. For very high viscosity ratio, viscoelasticity can induce drop migration towards the wall.

[1]Partially supported by NSF.

2:34PM R8.00009 Significance of viscoelastic effects on the rising of a bubble and bubble-to-bubble interaction , ARTURO FERNANDEZ, North Carolina A&T State University — Numerical results for the rising of a bubble and the interaction between two bubbles in non-Newtonian fluids will be discussed. The computations are carried out using a multiscale method combining front-tracking with Brownian dynamics simulations. The evaluation of the material properties for the non-Newtonian fluid will be discussed firstly. The results from the computations of a single bubble show how elastic effects modify the deformation and rising of the bubble by pulling the tail of it. The relationship between the strength of the elastic forces and the discontinuity in the bubble terminal velocity, when plotted versus bubble volume, is also observed in the computations. The bubble-to-bubble interaction is dominated not only by elastic effects but also by the shear-thinning caused by the leading bubble, which leads the trailing bubble to accelerate faster and coalesce with the leading bubble.

2:47PM R8.00010 Viscoelastic droplet deformation in complex flow: application of the extended finite element , ARASH SARHANGI FARD, MARTIEN HULSEN, PATRICK ANDERSON, Eindhoven University of Technology — Deformation of a viscoelastic drop in a Newtonian matrix under Stokes flow is simulated using a eXtended finite element method (XFEM). We consider the matrix inside a continuous mixer (like twin screw extruder), where shear and elongational flows are presents. Surface tension is included in the traction vector across the interface between the droplet and the matrix. The governing balance equations are solved once over entire domain (droplet and matrix). The surface of droplet is described by a discretized mesh and its position is updated by tracking the nodal positions in time. For XFEM integration, the position of the surface is determined explicitly by a numerical level set. To couple the velocity between droplet and matrix, a constrain is applied on balance equations. Constraints are enforced using a Lagrangian multiplier and also using weak interface conditions. Results are discussed for different viscosity ratio's and different Weissenberg numbers.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R10 General Fluid Dynamics III 313

12:50PM R10.00001 An Investigation of Length Scales in Active Grid Generated Turbulence[1] , R. JASON HEARST, PHILIPPE LAVOIE, University of Toronto — A novel active grid design is used to generate high Reynolds number homogeneous, isotropic turbulence. Typically, active grids consist of square wings mounted on a bi- planar mesh of rods. Each rod of the mesh is controlled independently by a stepper motor, which receives randomly varied signals defining its period and speed of rotation. The present design consists of two bi-planar rod meshes. Wing placement alternates between the forward and aft meshes, allowing for the motion of adjacent wings to be decoupled by mounting them on independently rotating rods. By changing the degree of correlation between the motion of wings, the length scales of the output turbulence may be influenced while the turbulence itself remains approximately homogeneous and isotropic. Furthermore, the structure of the turbulence may be dependent on the wing geometry. As such, alternative wing geometries (e.g. circular) are also investigated. The primary focus of this study is to investigate the effect of initial conditions on the length scales of turbulence in the context of active grid generated turbulence.

[1]This research is funded by an NSERC Engage award.

1:03PM R10.00002 Identification and manipulation of dynamic modes in nematic liquid crystals[1] , ZRINKA GREGURIC FERENCEK, TYRUS BERRY, TIMOTHY SAUER, JOHN ROBERT CRESSMAN, George Mason University — We present work identifying and manipulating patterns formed in an electroconvecting nematic liquid crystal. The existence of coherent, temporally and spatially correlated, structures are found in a wide range of driven systems. The development of new experimental and analytical techniques has enabled the identification of these structures and is beginning to elucidate their role in establishing macroscopic behavior. Here we describe an algorithm used to identify and track these structures, and report on the effects of local and global stimulation on their creation and evolution.

[1]EFRI-1024713

1:16PM R10.00003 Power consumption and dynamic structure in electroconvecting liquid crystals[1] , JOHN CRESSMAN, ZRINKA GREGURIC FERENCEK, TYRUS BERRY, TIMOTHY SAUER, George Mason University — A wide range of driven systems display complex, but correlated, dynamics as well as large fluctuations in energy injection, storage, and dissipation. We use an electroconvecting nematic liquid crystal as a model system to investigate the relationship between dynamical structure and energy flow. We use a dimensionality-reduction algorithm to identify the creation, evolution and annihilation of patterns of defects in a weakly-driven electroconvective state. By simultaneously measuring the electrical power drawn by the sample we are able to determine correlations between energy injection and defect dynamics. We will discuss these correlations as well as the interplay between energy flow and dynamic structures in this system.

[1]NSF grant EFRI-1024713

1:29PM R10.00004 Hot balls splash and sink fast , JEREMY MARSTON, King Abdullah University of Science and Technology, IVAN VAKARELSKI, SIGURDUR THORODDSEN, KAUST, DEREK CHAN, University of Melbourne — When a heated sphere is immersed in a liquid, we induce an inverted Leidenfrost effect whereby the sphere is wrapped in a vapour jacket which protects it from physical contact with the liquid and, when released to fall freely in the liquid, the sphere's terminal velocity can increase dramatically compared to a cold ball. This Leidenfrost-induced vapour layer can lead to significant drag reduction by up to 85% which appears to be the limiting case for drag reduction techniques based on gas layer injection. In a related experiment, when the heated sphere is released from above the surface, the dynamics of the entry are significantly different from the cold case, resulting in a prompt splash and cavity formation. We propose that this experiment is the ultimate non-wetting scenario during water-entry problems.

1:42PM R10.00005 Spatial characterization of underwater turbine wakes using Three-dimensional three-component (3D3C) velocity measurements , SEUNG-JAE LEE, LEONARDO CHAMORRO, University of Minnesota, DAN TROOLIN, TSI inc., ROGER ARNDT, FOTIS SOTIROPOULOS, University of Minnesota — A growing interest in underwater turbines (using tidal, river, and marine currents among others) has been observed during the last few years. A fundamental understanding of the turbulent flow around underwater turbines is crucial to predicting the potential effects of these structures on the local morphology, water flow, and power available in the current. The flow structure downstream of a turbine are inherently complex in that it is both unsteady and three dimensional. For this reason, a measurement technique that captures the full volumetric three-dimensional velocity field is advantageous for analyzing the complexity of the flow. In this study, a miniature horizontal axis 3-blade underwater turbine, aligned in the direction of the mean flow, was placed in a water flume at the St. Anthony Falls Laboratory at the University of Minnesota. Three-dimensional three-component (3D3C) velocity measurements were made in the flow downstream of the miniature underwater turbine. Wake evolution, turbulence characteristics and primary three-dimensional flow features were identified in the instantaneous and ensemble-averaged flows at different locations downstream of the turbine.

1:55PM R10.00006 Physics of badminton shuttlecock. Part 2 : Turn around , BAPTISTE DARBOIS TEXIER, CAROLINE COHEN, DAVID QUE RE, CHRISTOPHE CLANET, LadHyX, Ecole Polytechnique — We study experimentally shuttlecocks dynamics. In this part we show how the shuttlecock shape is optimized for badminton play. The shuttlecock always flies the nose forehead. After the impact it has thus to return. Actually it returns, oscillates and then stabilizes. We try to understand these damping oscillations and draw an analogy with the dandelions achenes and parachutists.

2:08PM R10.00007 Laminar jet injection in a pipe with co-flow , JAIME SCHMIEG, MARK STREMLER, PAVLOS VLACHOS, Virginia Tech — Particle Tracking Velocimetry (PTV) was used to investigate confined injection from a generic end-hole catheter within an axial co-flow environment. PTV was carried out in an iterative fashion where an approximate field was calculated from several time instances, and subsequently used as an estimator for hybrid tracking. The influence of momentum ratio on jet expansion, transport, and flow patterns was studied for several velocity ratios (VR $= \mathrm{V}_{jet}/\mathrm{V}_{out}$) between 0 and 10. The Reynolds number of the outer flow was 150 and that of the inner flow varied from 0 to 260. Flow patterns behind the catheter were dependent upon VR with recirculation regions present for low ratios. A separation bubble was observed behind the catheter for velocity ratios below 0.5 and two counter-rotating vortices were seen for VR = 0. As VR increased, asymmetry in the outer flow resulted in a single vortex behind the catheter with its position skewed toward the low flow side of the vessel and larger entrainment was present on the high flow side of the vessel. As VR increased above 0.5, recirculation was not observed and at VR = 1.0 the jet velocities were mainly in the streamwise direction.

2:21PM R10.00008 Effects of Exhaust Gas Recirculation on SI Engines at Wide Open Throttle , SYDNEY BRONSON, Carroll College, PAULIUS PUZINAUSKAS, University of Alabama — Exhaust gas recirculation, a charge dilution technique, has proven to be an effective method of reducing NOx emissions and fuel consumption of spark ignition engines. Wide open throttle operation also increases overall engine efficiency by reducing the pumping losses caused by throttling. In this study, the emissions and fuel economy benefits of exhaust gas recirculation (EGR) at wide open throttle conditions were quantified using a 2.4L port-injected engine. Engine performance and emissions data were recorded as the percentage of EGR in the intake charge was increased from zero to just above thirty percent (the EGR limit). This EGR percentage, in-cylinder pressure measurements, and the temperatures and pressures of the intake and exhaust were all recorded to ensure stable operating conditions. These tests were performed with a stoichiometric air-fuel ratio at a constant speed of 2000 rpm at wide open throttle. The variation of brake specific fuel consumption and emissions (in particular NOx) with increasing EGR percentages was analyzed.

2:34PM R10.00009 Pressure-velocity correlations in a flow upstream of a forward-facing step , DAVID PEARSON, Imperial College London, PAUL GOULART, ETH Zurich, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton, IMPERIAL COLLEGE FLOW CONTROL GROUP TEAM — The 2-dimensional velocity field upstream of a forward step was determined experimentally using Particle Image Velocimetry. A total of 4 seconds of data was acquired at $8000\mathrm{Hz}$. The flow velocity was $10\mathrm{ms}^{-1}$ with an $\mathrm{Re_h}$ of 20000, where $h = 0.03\mathrm{m}$ is the step height. The boundary layer thickness relative to step height was $\delta/h = 1.6$. The upstream surface pressure fluctuations were simultaneously measured using an array of 9 microphones embedded in tunnel floor. These pressure fluctuations are shown to have a direct linear correlation to the velocity perturbations. The correlation has a maximum of approximately 0.3 at upstream stations $x/h > 2$ and reduces toward background noise levels as the flow approaches separation at $0.5 < x/h < 1.5$. It is also shown that large pressure fluctuations upstream correlate to changes in shape and structure of the separation region at the step. This data demonstrates the ability to estimate some flow characteristics at the step face from the oncoming boundary layer, through the use of pressure measurements at the wall.

2:47PM R10.00010 ABSTRACT WITHDRAWN —

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R11 Turbulence: Shear Layers II 314

12:50PM R11.00001 Scaling Law for the Onset of Turbulence in Channel Flow , GRÉGOIRE LEMOULT, JEAN-LUC AIDER, JOSE EDUARDO WESFREID, PMMH (ESPCI-CNRS) Paris, France — We are presenting experiments about the onset of turbulence in channel flow. We perturbed the base plane Poiseuille flow with continuous injection of water normally to the flow. We performed Particle Image Velocimetry experiments to measure the mean velocity profile with respect to the Reynolds number, Re and the amplitude of the perturbation u_{jet}/u_{cl}. Due to the unstable nature of the flow with respect to finite amplitude perturbation, it should exist a minimal amplitude ε, which trigger the transition to the turbulence. In order to find this critical value, we define a new experimental criterion using the value of $\tilde{u} = \bar{u}_{cl}/\bar{u}_{cl,unperturbed}$, giving the deformation of the mean velocity profile. We found a power law for the onset of turbulence $\varepsilon = O(Re^{-3/2})$ and compared it with different models and previous experiments.

1:03PM R11.00002 Experimental Evolution of Local and Global Variables in the Subcritical Transition in Channel Flow , JOSE EDUARDO WESFREID, GREGOIRE LEMOULT, JEAN-LUC AIDER, PMMH (ESPCI-CNRS) Paris, France — We perform experiments on the subcritical transition to the turbulence in a water channel with plane Poiseuille flow, perturbed by controlled injection of water normally to the wall. For different values of Reynolds number Re and different amplitude of the perturbation u_{jet}/u_{cl}, we observed different states from laminar to turbulent. Using Particle Image Velocimetry, we study the dynamics of a local variable of the velocity field as the transverse magnitude and simultaneously we follows a global one, as the deformation of the mean velocity profile $\tilde{u} = \bar{u}_{cl}/\bar{u}_{cl,unperturbed}$. We discuss the evolution in the phase space of those variables as a function of the strength of the perturbation, and compare it with predictions made from low dimensional models.

1:16PM R11.00003 The low-frequency undulation mechanism in a open cavity shear layer flow[1] , XIAOFENG LIU, JOSEPH KATZ, Johns Hopkins University — The unsteady flow and pressure field in an open cavity shear layer has been investigated using time resolved PIV measurements. Here, we focus on the closed-loop feedback mechanism that causes low-frequency undulation in the location of the shear layer relative to the cavity trailing corner. It is found that impingement of high momentum fluid on the forward-facing surface of this corner, when the shear layer is low, periodically increases the pressure there, and the inward flow along this wall. The latter re-circulates back to the leading corner of the cavity, increasing the pressure there, and imposing adverse pressure-gradients on the boundary layer upstream of the cavity. The resulting increase in boundary layer thickness starts an upward shift in the elevation of the shear layer, which reduces the momentum, pressure and jetting into the cavity at the downstream corner. This reduced backflow decreases the pressure at the inlet corner and the incoming boundary layer thickness, causing a downward shift in elevation of the shear layer. This low-frequency undulation has substantial impact on the turbulence and noise generations by the cavity corner.

[1]Sponsored by ONR and NSF.

1:29PM R11.00004 Simultaneous measurement of pipe flow downstream and upstream of 90 degrees bend by using stereo PIV[1] , JUN SAKAKIBARA, NOBUTERU MACHIDA, Department of Engineering Mechanics and Energy, University of Tsukuba — We measured velocity vector distribution in cross sections of a fully developed turbulent pipe flow upstream and downstream of a 90 degrees bend by using stereo PIV and single camera PIV simultaneously. Reynolds number was Re=27,000, and ratio of inner diameter d (=50mm) of the pipe and radius of the centerline of the bend was 1.0. Instantaneous flow field downstream of the bend represented the unsteady motion of the anti-symmetric counter-rotating Dean vortices. Stagnation point, which was created by the flow induced by the two vortices, was observed at both inner and outer side of the bend. The stagnation point moves unsteadily within roughly 30 degree above and below the symmetry plane. In order to clarify the origin of such unsteady motion of the stagnation point, conditional average of the upstream velocity vector field conditioned by the azimuthal location of the stagnation point downstream the bend was computed. Under condition where the inner side stagnation point stays above (below) the symmetry plane, the conditional streamwise velocity upstream the bend exhibited a positive peak below (above) the plane on the outer side.

[1]This study was partially supported by Tokyo Electric Power Company.

1:42PM R11.00005 ABSTRACT WITHDRAWN —

1:55PM R11.00006 The turbulent wake of a submarine model at varying pitch and yaw angle , MILOUD ALAOUI, ENSMA, ANAND ASHOK, ALEXANDER SMITS, Princeton University — The objective of the present study is to understand how the pitch and yaw angle affect the mean flow and turbulence in the wake of an axisymmetric submarine model (DARPA SUBOFF model). Measurements in the wake were performed at a Reynolds number based on the length of 2.37×10^6. Mean velocity and two-component turbulence measurements were performed using Pitot probes and hot wires in the span-wise plane at three different downstream positions: 5, 7.5 and 10 diameters downstream of the trailing edge. The range of measured angles of attack and yaw angles were limited to between 0 and 10° in part to avoid wind tunnel interference effects. Work supported by ONR Grant N00014-09-1-0263.

2:08PM R11.00007 Mixing length and scale-to-scale kinetic energy transfer in the wake of a fractal tree[1] , KUNLUN BAI, CHARLES MENEVEAU, JOSEPH KATZ, Mechanical Engineering and CEAFM, Johns Hopkins University — To study the dynamics of turbulence interacting with multi-scale objects, we measure turbulence structure in the wake of a fractal tree-like object in a water channel, using PIV. The eddy viscosity is obtained from the correlation of mean shear and Reynolds shear stress distributions across the wake. We address the question whether a mixing length-scale can be identified in this flow, and if so, how it relates to the geometric length-scales in the pre-fractal object. One approach is based on spectral distributions. Another more practical approach is based on length-scale distributions evaluated using fractal geometry tools. These models agree well with the measured mixing length indicating that information about multi-scale clustering of branches has to be taken into account. To explore the energy transfer at different scales, data are spatially filtered at various scales and the subgrid-scale flux among scales is evaluated. In contrast to regions characterized by a single length scale (bottom of the tree) where a classical inertial range cascade behavior is observed with scale-independent flux, at heights where multiple branch generations interact with the flow, we find that the energy flux depends strongly on the filter size, increasing with smaller filter sizes. The results can be explained by production at multiple-scales.

[1]Supported by NSF ATM-0621396 and AGS-1047550.

2:21PM R11.00008 Heated Jets Emitting From a Rectangular Stack into a Cross Wind , B.E. JOHNSON, G. ELLIOTT, K.T. CHRISTENSEN, University of Illinois — A detailed analysis of jets in cross-flow is performed where jets heated to a centerline temperature $T_\circ = 425\,\mathrm{K}$ emit from a rectangular stack ($AR = 3.76$) into $T_\infty = 300\,\mathrm{K}$ cross-wind at area-averaged momentum flux ratio $r = 3.3$. Cross-flow and jet centerline velocities are $U_\infty = 10\,\mathrm{m/s}$ and $V_\circ = 50\,\mathrm{m/s}$, respectively. Injection is normal to the bounding wall from a raised stack such that the initial incidence of jet interaction with cross-flow occurs well outside of the boundary layer. Rake-mounted thermocouple measurements of mean temperature and cross-plane stereoscopic PIV measurements of the turbulent flow field are performed at multiple stations downstream of the stack in spanwise–wall- normal regions of interest. Stack yaw angles of 0°, 45°, and 90° comprise a set of key orientations from which inferences can be made of real-world heated jet in cross-flow behavior where cross-flow directionality may vary under shifting mean wind direction. From the measurements made under each of these stack orientations, the downstream dispersion of the heated jet fluid is characterized as is the downstream evolution of the turbulence and associated vortical structures.

2:34PM R11.00009 Turbulent structures of impinging circular jet , VESSELINA ROUSSINOVA, RAM BALACHANDAR, University of Windsor — Turbulent impinging jets are used in various engineering applications due to their ability to provide superior heat and mass transfer. In hydraulic engineering, impinging jet flows have a detrimental effect due to their ability to scour and erode sand beds. In order to gain a better insight into the mean flow, turbulence and coherent structures in impinging jet flows, we performed high resolution particle image velocimetry (PIV) measurements of a round normally impinging jet issuing from a nozzle with diameter d = 0.01 m at Reynolds number Re =20 000 and at the jet-to-plate distance H = 20d. This configuration was chosen to match previously reported experiments and to verify results obtained from numerical simulations in which several phenomena have been noted, but the underlying turbulence dynamics remained obscure. PIV velocity fields are measured in the streamwise - spanwise (x-z) planes in the free jet and stagnation regions while streamwise - wall normal (x-y) planes are probed in the radial wall jet region in the immediate proximity of the impinging plate. The focus of this study is to investigate in details mean velocities, various turbulent quantities and vorticity. Analysis of the coherent structures is also documented through the analysis of swirling strength and proper orthogonal decomposition (POD).

2:47PM R11.00010 Half-loop and full-loop shedding in the wake of wall-mounted square cylinders due to boundary layer-wake interaction , JASON BOURGEOIS, ZAHRA HOSSEINI, ROBERT MARTINUZZI, University of Calgary — The vortical flow in the finite wall-mounted obstacle wakes can be important in heat transfer devices, turbomachinery components, and flame stabilizer devices, and is of fundamental importance since it displays fully three-dimensional states of Kármán vortex streets that are found in 2D bluff body wakes. The turbulent state of the wake of a finite square cylinder (height-to-width ratio h/w=8) has been found to be sensitive to the conditions of the approach boundary layer. The energetic quasi-periodic vortical structure topology is found to change between two topological states. Boundary layer thicknesses δ/d=0.7 and 1.6 yield half-loop and full-loop structures, respectively. This modification of the structure topology has significant consequences for modifying the mixing, momentum transfer, turbulence production, and Reynolds stresses in the wake. Using synchronized particle image velocimetry (PIV) and surface pressure measurements for these two boundary layers, the coherent structures in the wake of the cylinder are reconstructed and analyzed. Vortical connector strands which tie together subsequently shed structures lead to high incoherent Reynolds stresses, streamwise vorticity, vortex stretching, and turbulence production in their neighbourhood, but do not appear in the lower regions of the wake for the half-loop topology.

Tuesday, November 22, 2011 12:50PM - 3:00PM –

Session R13 Multiphase Flow VII 316

12:50PM R13.00001 Modeling of Sulfuric Acid Condensation on Heat Exchanger Cooling Fins , XIAOBAI LI, DAVID COOK, Robert Bosch LLC Research and Technology Center-North America — Sulfuric acid corrosion on metallic heat exchanger cooling fins can cause serious blockage problem and stop the normal operation of heat exchangers. Corrosion rates are strongly dependent on surface film pH value. Therefore, a multi-physics computational framework was developed to predict the liquid film formed on solid surface and the pH distribution. Such a model can be used for better understanding of acid condensation from multi-species system. In this work, first, from S to H_2SO_4, formation of sulfuric acid in gas phase during combustion and cooling process was investigated with detailed chemistry mechanisms. The amount of SO_2 and SO_3 that plays important role in acid condensation process was calculated. Then, multi-component condensation process was modeled to produce a liquid film of acid and water solution condensed on solid surface that has low temperature. pH value was obtained based on the concentration of the acid. The above work provides critical information for corrosion analysis for heat exchangers.

1:03PM R13.00002 Reduced model for multi-component condensation , DENNIS VAN PUTTEN, ROB HAGMEIJER, University of Twente, RYAN SIDIN, University of Suriname — An extremely efficient reduced model for multi-component condensation is presented. The Becker-Döring (BD) equations are approximated by the General Dynamic Equation (GDE) in the supercritical region. The subcritical and critical regions are replaced by a source point that injects clusters into the supercritical region. The location and strength of the source point are determined from local estimates of the Gibbs free energy function, and the diffusion term in a corresponding Fokker-Planck equation. The result is a curve in composition space with Dirac delta function character in planes perpendicular to the curve. Integral properties of the GDE and BD solutions compare well for a typical two-component nucleation pulse experiment and computational effort is reduced by five orders of magnitude.

1:16PM R13.00003 Modeling of moving contact lines on electrically charged heated surfaces , VLADIMIR AJAEV, Southern Methodist University — Local fluid flow and heat transfer near moving contact lines on heated surfaces is usually described by mathematical models incorporating the effects of evaporation, surface tension, thermocapillarity, and disjoining pressure due to London-van der Waals forces. However, this description is not accurate for the cases when electric charges in the liquid and on the heated surface are present, which is usually the case in applications involving water and liquid metals. We develop a model which incorporates the electrostatic effects into the standard lubrication-type model of a contact line on a heated surface. The gas phase above the liquid is assumed to be pure vapor and the macroscopically dry region of the solid is covered with an adsorbed film. The local liquid-vapor interface shapes and the apparent contact angle are described as functions of the temperature and the charge density at the solid surface.

1:29PM R13.00004 Sustained inertial-capillary oscillations and jet formation in displacement flow in a tube , YI SUI, Department of Chemical Engineering, Imperial College London, UK, PETER D.M. SPELT, LMFA, CNRS and Universite Claude Bernard Lyon 1, France — We study inertial effects in the displacement of a fluid in a capillary by a more viscous fluid using a level-set method. Various flow regimes are identified with a Reynolds and a capillary number as the main parameters. At relatively low Reynolds number, the meniscus forms a steady shape, and the interfacial curvature at the tube centre could change from being concave to convex upon increasing the Reynolds number. Beyond a critical Reynolds number, a quasi-steady solution is no longer found for sufficiently small contact angle values (less than 80 degrees): instead, the interface undergoes non-dampened periodic oscillations and, at even larger values of the Reynolds number, quasi-periodically, and the interface evolves from simple wavy shapes to complex shapes with multiple wavy units. Beyond a second critical Reynolds number, the liquid forms a jet and the meniscus advances with a nearly constant speed which decreases with Re. This is also observed at large contact angle values. In a developing jet, however, the interface shape remains partially quasi-steady, near the contact line region and the tube centre. The flow behaviour is shown to be robust over a range of other governing parameters, including the capillary number and the slip length.

1:42PM R13.00005 Bubbles in an isotropic homogeneous turbulent flow , F.E. MANCILLA, M. MARTINEZ, E. SOTO, G. ASCANIO, R. ZENIT, Universidad Nacional Autonoma de Mexico — Bubbly turbulent flow plays an important role in many engineering applications and natural phenomena. In this kind of flows the bubbles are dispersed in a turbulent flow and they interact with the turbulent structures. The present study focuses on the motion and hydrodynamic interaction of a single bubble in a turbulent environment. In most previous studies, the effect of bubbles on the carrier fluid was analyzed, under the assumption that the bubble size was significantly smaller that the smallest turbulence length scale. An experimental study of the effect of an isotropic and homogeneous turbulent flow on the bubble shape and motion was conducted. Experiments were performed in an isotropic turbulent chamber with nearly zero mean flow, in which a single bubble was injected. The fluid velocity was measured using the Particle Image Velocimetry (PIV) technique. The bubble deformation was determined by video processing of high-speed movies. The fluid disturbances on the bubble shape were studied for bubbles with different sizes. We will present experimental data obtained and discuss the differences among these results to try to understand the bubble - turbulence interaction mechanisms.

1:55PM R13.00006 Response of floating grains on a capillary Faraday wave in the dense limit , CEYDA SANLI, DEVARAJ VAN DER MEER, DETLEF LOHSE, Physics of Fluids, University of Twente, The Netherlands — When macroscopic grains float on a water-air interface, they aggregate into a static cluster due to attractive gravity-induced capillary interaction between the grains. Here we study what happens when the grains in the cluster are excited using capillary Faraday waves with a wavelength comparable to the grain size. The grains are found to exhibit cooperative motion in domains which tend to become larger as the particle concentration ϕ increases towards the jamming point. We measure the average grain velocity and the velocity-velocity correlations of the grains in space and time as a function of ϕ and the driving amplitude a.

2:08PM R13.00007 Breakup of particle clumps on liquid surfaces , S. GURUPATHAM, M. HOSSAIN, B. DALAL, I. FISCHER, P. SINGH, New Jersey Institute of Technology, D. JOSEPH, University of Minnesota — In this talk we describe the mechanism by which clumps of some powdered materials breakup and disperse on a liquid surface to form a monolayer of particles. We show that a clump breaks up because when particles on its outer periphery come in contact with the liquid surface they are pulled into the interface by the vertical component of capillary force overcoming the cohesive forces which keep them attached, and then these particles move away from the clump. In some cases, the clump itself is broken into smaller pieces and then these smaller pieces break apart by the aforementioned mechanism. The newly-adsorbed particles move away from the clump, and each other, because when particles are adsorbed on a liquid surface they cause a flow on the interface away from themselves. This flow may also cause particles newly-exposed on the outer periphery of the clump to break away. Since millimeter-sized clumps can breakup and spread on a liquid surface within a few seconds, their behavior appears to be similar to that of some liquid drops which can spontaneously disperse on solid surfaces.

2:21PM R13.00008 Surface shear viscosity effects on the damping of oscillations in millimetric liquid bridges , MIGUEL A. HERRADA, Escuela Técnica Superior de Ingenieros, Universidad de Sevilla, E-41092-Sevilla, Spain, JOSÉ M. MONTANERO, Escuela de Ingenieros Industriales, Universidad de Extremadura, E-06006-Badajoz, Spain, JOSÉ M. VEGA, E.T.S.I. Aeronáuticos, Universidad Politécnica de Madrid, 28040-Madrid, Spain — The damping rate of the small free oscillations in a non-cylindrical, axi-symmetric liquid bridge between two circular disks is calculated and compared with some previous experimental measurements using hexadecane in a millimetric liquid bridge. Current theories, accounting for viscous damping in both the boundary layers attached to the disks and the bulk, underestimated the measured damping by a $O(1)$ quantity. Calculations based on the full Navier-Stokes equations are also in disagreement with the experimental results. These discrepancies are essentially eliminated in this work considering the effect of the surface shear viscosity (whose value results from empirical fitness), which could be due to the presence of a contaminating monolayer. Some conclusions are extracted in connection with surface wave damping in micro-fluidic devices.

2:34PM R13.00009 Optimized evaporation from a microchannel heat sink , REZA MONAZAMI, HOSSEIN HAJ-HARIRI, University of Virginia — Two-phase heat transfer devices, benefiting the unique thermal capacities of phase- change, are considered as the top choice for a wide range of applications involving cooling and temperature control. Evaporation and condensation in these devices usually take place on porous structures. It is widely accepted that they improve the evaporation rates and the overall performance of the device. The liquid menisci formed on the pores of a porous material can be viewed as the active sites of evaporation. Therefore, quantifying the rate of evaporation from a single pore can be used to calculate the total evaporation taking place in the evaporator given the density and the average size of the pores. A microchannel heat sink can be viewed as an structured porous material. In this work, an analytical model is developed to predict the evaporation rate from a liquid meniscus enclosed in a microchannel. The effects of the wall superheat and the width of the channel on the evaporation profile through the meniscus are studied. The results suggest that there is an optimum size for the width of the channel in order to maximize the thermal energy absorbed by the unit area of the heat sink as an array of microchannels.

2:47PM R13.00010 Numerical simulation of heat transfer provided by an impinging droplet train[1] , STEVEN R. LEWIS, MARIO F. TRUJILLO, University of Wisconsin, Madison — A detailed investigation of the parameters that affect cooling within the thermal boundary layer created by a stream of impinging HFE-7100 droplets striking a pre-wetted and heated surface is performed. After the initial transient has ended, the flow enters a quasi-steady state in which the liquid crown formed during continuous droplet impact remains nearly stationary. Factors including initial film thickness, surface tension, droplet velocity, volumetric flow-rate and droplet frequency are categorized as either contributing to changing the thickness of the thermal boundary layer or as non-contributing parameters. Additionally, an analytical solution for the growth of the thermal boundary layer is proposed, using a crown propagation model, to describe the flow within the boundary layer. The analytical model shows good agreement with numerical results and incorporates the influence of the previously identified parameters.

[1]The authors are grateful for the support from the Office of Naval Research, code 331, and to Mark Spector, its program director.

Tuesday, November 22, 2011 12:50PM - 2:34PM –
Session R14 Experimental Techniques V 317

12:50PM R14.00001 Velocity measurements in a thermoacoustic refrigerator using Time-Resolved Particle Image Velocimetry , PHILIPPE BLANC-BENON, LMFA, UMR 5509, Ecole Centrale Lyon, France, GAELLE POIGNAND, LAUM UMR CNRS 6613, ARGANTHAEL BERSON, Durham University, EMMANUEL JONDEAU, LMFA, UMR CNRS 5509, Ecole Centrale Lyon — A standing-wave thermoacoustic refrigerator consists of a stack of plates placed in an acoustic resonator with two heat exchangers located at each end of the stack. The full understanding of the heat transfer between the stack and the heat exchangers of thermoacoustic systems is a key issue to improve the global efficiency of such devices. The aim of this work is to investigate the generation of vortices near the ends of the stack, which affects heat transfer. The aerodynamic field in the gap between the stack and the heat exchanger is characterized using a time-resolved particle image velocimetry technique. Measurements are performed in a standing-wave refrigerator operating at a frequency of 200 Hz. Instantaneous velocity fields are recorded at a frequency of 3125 Hz (i.e. 15 velocity fields per acoustic period). Measurements show that vortex shedding occurs at high pressure levels, when the nonlinear acoustic regime prevails and they validate previous experiments [Berson & Blanc-Benon, J. Acoust. Soc. Am., 2007, 122(4), EL122-127]. The increased viscous dissipation generates additional heating and a loss of efficiency.

1:03PM R14.00002 Error Reduction in Molecular Tagging Velocimetry (MTV) Processing Using Image Filtering[1] , MICHAEL CASO, DOUGLAS BOHL, Clarkson University — Prior work has shown that the error level in MTV measurements is closely tied to the image SN level. In practice the SN ratio will depend on experimental conditions such as attenuation, Field of View, laser power, camera, etc.; however, there is a minimum SN level that can be achieved for any given experiment. Experience has shown that MTV images typically have a SN=2-8. It is therefore desirable to be able to lower image noise after the images are acquired to reduce measurement error. In this work post processing MTV images using image filtering schemes such as Gaussian Blur, FFT (band pass), median filtering etc. was investigated using synthetic MTV images with added random noise. The synthetic images were filtered and then processed using a direct correlation technique. The results showed that for very noisy images (i.e. SN<4) the all filtering techniques improved the displacement error by 10-40%. As the SN increased filtering because less effective in decreasing error and in some cases increased the measurement error. The FFT band pass filter was most effective and improved measurement error for all SN levels.

[1]Research Supported by NSF Grant #0845882.

1:16PM R14.00003 Determining the Shock Hugoniot of Transparent Materials with Hydrodynamic Pressure Loading , FORREST SVINGALA, GARY SETTLES, Penn State Gas Dynamics Lab — The shock Hugoniot is a fundamental relationship between pressure, volume, and energy for a given material. Accurate knowledge of the Hugoniot for a material is critical in order to determine its response to blast waves and ballistic impacts. Traditionally, the shock Hugoniot is measured on a point-by-point basis through an extensive series of high-velocity impact experiments. Observations are confined to pointwise pressure or velocity measurements at the free surfaces of the sample. In this work a new technique is presented, one which allows multiple points of the shock Hugoniot to be determined in a single experiment. A gram-scale explosive charge is detonated to produce an unsteady shock wave in the transparent material sample. Pressure between the charge and sample is initially high, but is rapidly reduced by expansion of the explosive product gases. This loading produces an initially strong shock wave, which attenuates to near the bulk sound speed as it transits the sample. Using a high-speed shadowgraph technique, multiple shock and particle velocity combinations are observed in a single experiment. This allows the measurement of a shock Hugoniot in fewer experiments than by traditional methods. This technique produces data in agreement with published Hugoniot results for polyurethane. It can be easily extended to measure the Hugoniot of any transparent solid, liquid, or gas.

1:29PM R14.00004 Toward an automated background oriented schlieren (BOS) system , MICHAEL HARGATHER, Gas Dynamics Laboratory, Penn State University, GARY SETTLES[1], Penn State University — The background oriented schlieren (BOS) technique is a useful method for visualizing refractive disturbances in a wide range of experimental settings. The technique visualizes refractive disturbances via their distortion of a distant background pattern (typically a speckle pattern). A cross-correlation computer algorithm is typically used to identify and measure distortions of the background pattern, thus revealing the refractive disturbance changes between images and producing a schlieren image. The cross-correlation algorithm, however, can be time-consuming and prevents an instantaneous schlieren image from being observed, thus hampering some potential BOS applications. Here a novel background patterning approach is presented which eliminates the need for the cross-correlation algorithm. Results are presented showing the sensitivity of the new background pattern and its potential application for providing instantaneous BOS images. Background pattern characteristics are explored for high- and low-speed fluid-dynamic applications.

[1]Gas Dynamics Laboratory, Penn State University

1:42PM R14.00005 Molecular candidates of MTV in air , NICO DAM, Mechanical Engineering Department, Eindhoven University of Technology, PO Box 513, 5600 MB Eindhoven, the Netherlands, MEHRNOOSH MIRZAEI, Institute for Molecules and Materials, Radboud University of Nijmegen, PO Box 9010/51. 6500 GL Nijmegen, the Netherlands, WILLEM VAN DE WATER, Physics Department, Eindhoven University of Technology, PO Box 513, 5600 MB Eindhoven, the Netherlands — In molecular tagging velocimetry (MTV), the molecules of a gas are used as flow tracers. These tracers can be produced at will by illumination with a laser which promotes molecules to a long- lived excited state, fuses N_2 and N_2 to NO, or makes molecules phosphoresce. A while later these tagged molecules can be visualized by laser-induced fluorescence, or by just watching them while they phosphoresce. Candidates for MTV in turbulence research must be arranged in structures narrower than the Kolmogorov scale, which remain narrow as time progresses, and must live longer than the Kolmogorov time. These requirements invalidate many candidates, candidates once deemed successful. They do so in various surprising manners that involve a combination of fluid flow and molecular dynamics. Rather than velocimetry in turbulence, MTV techniques offer a unique view on basic dispersion processes at the smallest scales of turbulence. 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.

1:55PM R14.00006 Constant Current Plasma Anemometer[1] , CURTIS MARSHAL, ERIC MATLIS, THOMAS CORKE, University of Notre Dame, SIVARAM GOGINENI, Spectral Energies, LLC — An improved design for a plasma anemometer that provides mega-Hertz bandwidth velocity measurements over a range of Mach numbers from subsonic to hypersonic, is presented. The anemometer uses a small volume of ionized air as the sensor. The ionized air is formed between two electrodes that are powered by a high-frequency AC voltage source. Resistance and capacitance elements in the AC power circuit that simulate a dielectric-barrier, have been added to prevent transient filament formation. This resulted in a 300-times reduction in the anemometer EMI of previous designs. In addition, a closed-loop feedback control has been added to maintain a constant current through the sensor. With constant current operation, the voltage drop across the ionized air between the electrodes varies linearly with air velocity (or mass-flux in a compressible flow). This is demonstrated through mean velocity calibrations of the anemometer for a range of velocities and feedback parameters. The dynamic response and other capabilities of the sensor will also be demonstrated.

[1]Supported under NASA SBIR Program

2:08PM R14.00007 In-situ shear stress indicator using heated strain gages at the flow boundary , CHI-AN YEH, FULING YANG, ME, Natl Taiwan Univ — This work borrows the concept of hot-wire anemometry and sketch a technique that uses local heat transfer to infer the flow field and the corresponding stress. Conventional strain gages were mounted at the flow solid boundary as the heat source and acrylic boundary was chosen for its low thermal conductivity ensuring heat accumulation when a gage is energized. The gage would now work in slightly overheated state and its self-heating leads to an additional thermal strain. When exposed to a flow field, heat is brought away by local forced convection, resulting in deviations in gage signal from that developed in quiescent liquid. We have developed a facility to achieve synchronous gage measurements at different locations on a solid boundary. Three steady flow motions were considered: circular Couette flow, rectilinear uniform flow, and rectilinear oscillating flow. Preliminary tests show the gage reading does respond to the imposed flow through thermal effects and greater deviation was measured in flows of higher shear strain rates. The correlation between the gage signals and the imposed flow field is further examined by theoretical analysis. We also introduced a second solid boundary to the vicinity of the gage in the two rectilinear flows. The gage readings demonstrate rises in its magnitudes indicating wall amplification effect on the local shear strain, agreeing to the drag augmentation by a second solid boundary reported in many multiphase flow literatures.

2:21PM R14.00008 Assessment of the Derivative-Moment Transformation method for unsteady-load estimation , ALI MOHEBBIAN, DAVID RIVAL, University of Calgary — It is often difficult, if not impossible, to measure the aerodynamic or hydrodynamic forces on a moving body. For this reason, a classical control-volume technique is typically applied to extract the unsteady forces instead. However, measuring the acceleration term within the volume of interest using PIV can be limited by optical access, reflections as well as shadows. Therefore in this study an alternative approach, termed the Derivative-Moment Transformation (DMT) method, is introduced and tested on a synthetic data set produced using numerical simulations. The test case involves the unsteady loading of a flat plate in a two-dimensional, laminar periodic gust. The results suggest that the DMT method can accurately predict the acceleration term so long as appropriate spatial and temporal resolutions are maintained. The major deficiency was found to be the determination of pressure in the wake. The effect of control-volume size was investigated suggesting that smaller domains work best by minimizing the associated error with the pressure field. When increasing the control-volume size, the number of calculations necessary for the pressure-gradient integration increases, in turn substantially increasing the error propagation.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R16 Porous Media III 319

12:50PM R16.00001 Mathematical modelling of membrane separation , FRANK VINTHER, Technical University of Denmark — Membrane separation is commonly used in chemistry and chemical engineering, where the separation of one or several species of molecules is of interest. This presentation will presents mathematical modelling of the dynamic interplay between the transport equations through the membrane and the transport equations within the bulk solution. Thus, resulting in a system of PDE's with time varying boundary conditions. The model is used for predicting optimal parameters for separation processes.

1:03PM R16.00002 Reynolds number dependent flow regime characteristics for flow in porous media[1] , JAMES LIBURDY, VISHAL PATIL, Oregon State University — The flow characteristics of distinct flow regimes in porous media and the evolution with Reynolds number is poorly understood at high Reynolds numbers. Typical means of measurement are MRI imaging, PTV and PIV, the latter two require refractive index matching. This study presents PIV measurements in a porous bed of spherical beads, for pore Reynolds numbers from 100 to 1500 resulting in flow in the inertial, unsteady inertial and turbulent flow regimes. The bed is a cube with five bead diameters on a side, with 15 mm beads. Measurements are based on two dimensional slices (five along the optical axis) using two fields of view; the first is four beads by four beads, and the second is of individual pores to provide highly spatial resolution. Fluorescent dye studies are presented. Velocity data are analyzed based on statistical results of two dimensional time series vector fields with emphasis on (i) identification of Reynolds number dependent flow structures (spatial and temporal), (ii) delineation of flow regime transitions, (iii) establishing pore-based fluctuation energy budgets and (iv) illustration of local dispersion characteristics. The goal is to provide statistical flow structure characteristics at the pore level for high Reynolds number flows to better understand dispersion characteristics.

[1]This study was supported in part by NSF through grant 0933857

1:16PM R16.00003 Pressure driven flow in porous tubular membranes , NILS TILTON, University of Maryland, DENIS MARTINAND, ERIC SERRE, University of Aix-Marseille, RICHARD LUEPTOW, Northwestern University — We consider the steady laminar flow of a Newtonian incompressible fluid in a porous tubular membrane with pressure-driven transmembrane flow. Due to its fundamental importance to membrane filtration systems, this flow has been studied extensively both analytically and numerically, yet a robust analytic solution has not been found. The problem is challenging due to the coupling between the transmembrane pressure and velocity with the simultaneous coupling between the axial pressure gradient and the axial velocity. We present a robust analytical solution which incorporates Darcy's law on the membrane surface. The solution is in the form of an asymptotic expansion about a small parameter related to the membrane permeability. We verify the analytical solution with comparison to 2-D spectral direct numerical simulations of ultrafiltration and microfiltration systems with typical operating conditions, as well as extreme cases of cross-flow reversal and axial flow exhaustion. In all cases, the agreement between the analytical and numerical results is excellent. Finally, we use the analytical and numerical results to provide guidelines about when common simplifying assumptions about the permeate flow may be made. Specifically, the assumptions of a parabolic axial velocity profile and uniform transmembrane velocity are valid only for small permeabilities.

1:29PM R16.00004 The Effect of Inlet Swirler Design on Passive Control of Combustion Noise and Instability[1] , ALEX BORSUK, State University of New York at Buffalo, AJAY AGRAWAL, JUSTIN WILLIAMS, University of Alabama — The use of porous inert media (PIM) in the reaction zone of a swirl-stabilized lean-premixed combustor provides a passive method of controlling combustion noise and instability. Swirl-stabilized combustors use an inlet swirler that imparts a swirling motion to the reactant flow and stabilizes the flame. In this study, the effect of swirler design and swirl number on combustion without and with PIM has been investigated experimentally, using a methane-fueled quartz combustor at atmospheric pressure. Swirler vane angle was varied to obtain swirl numbers of 0.45, 0.78, and 1.10. Swiler location was varied to obtain recess depth in the premixer tube of 0, 2.5, and 5 cm. Experiments were conducted at a constant air flow rate of 300 SLPM and equivalence ratios of 0.7, 0.75, and 0.8. PIM geometries with increasing and decreasing flow cross-sectional area were tested. The performance of each test case is compared by measuring sound pressure levels (SPL) with a microphone probe and observing the flame behavior. Results indicate that PIM can be effective in reducing noise and instability over a wide range of operating conditions. Total SPL reductions of up to 7.6 dBA were observed with PIM.

[1]Funded by NSF REU Site #10626110.

1:42PM R16.00005 How to Model the Lift Generation in a Highly Compressible Porous Media , Q. WU, R. NATHAN, S. SANTHANAM, T. GACKA, CBMSS TEAM — Lift generation in highly compressible porous media under rapid compression continues to be an important topic in porous media flow for its superior potential in soft lubrication and squeeze damping. Although significant progress has been made in the study of the lift generation experimentally and theoretically (Wu et al., Journal of Fluid Mechanics 542, 281, 2005; Barabadi, et al., Journal of Heat Transfer, 131(10), 101006, 2009), how to theoretically characterize lifting forces remains unclear. In this paper the permeability of the porous media was measured using a permeater, and then dynamically compacted in a porous-walled cylinder piston apparatus. The obtained pore pressure generation was compared to two different theoretical models, a plug flow model and a consolidation model used in Wu et al. (2005) and Barabadi, et al. (2009). It shows that the consolidation model is appropriate. Furthermore, a viscoelastic model, containing a nonlinear spring in conjunction with a linear viscoelastic Generalized Maxwell mechanical module, is developed to characterize the solid phase lifting force, showing excellent agreement with experimental data. The paper presented herein, conclusively demonstrates the validity of the theoretical approach developed by Wu et al. (2005) and provides a meaningful approach in characterizing forces that contribute to lift generation in soft porous media under rapid compression.

1:55PM R16.00006 From Red Cells to Soft Porous Lubrication , T. GACKA, CBMSS, R. NATHAN, L. WU, Q. WU, CBMSS LABORATORY TEAM, CHINESE ACADEMY OF SCI. TEAM — Feng and Weinbaum (*J. Fluid. Mech.*, **422**, 282, 2000), inspired by the enhanced lift phenomena in downhill skiing, developed a new lubrication theory for highly compressible porous media where significantly increased lifting force was predicted as a planing surface glided over a soft porous layer; suggesting superior potential use of porous media for soft lubrication. In this study, we experimentally examine the lift generation phenomena by developing a novel soft porous bearing that consists of a running conveyer belt covered with a soft, 100% polyester, porous sheet, and a stationary, fully instrumented, inclined, planar, upper board. Pore pressure was generated as the upper boundary glides over the soft porous bearing and was measured by pressure sensors. One observed that the pore pressure distribution is consistent with predictions by Feng and Weinbaum (2000), and is a function of the relative velocity between the planing surface and the running belt, the mechanical properties (e.g. porosity, permeability and stiffness) and thickness of the porous layer, as well as the compression ratios at the leading and trailing edges. A load cell is used to characterize the performance of the porous bearing, by comparing pore pressure to total lifting forces. The study presented herein significantly improves our understanding of the behavior of highly compressible porous media under fast compression.

2:08PM R16.00007 Probing the permeability of porous media by NMR measurement of stochastic dispersion dynamics , TYLER BROSTEN, ROBERT MAIER, Engineer Research and Development Center, U.S. Department of Defense, SARAH CODD, Department of Mechanical and Industrial Engineering, Montana State University, SARAH VOGT, JOSEPH SEYMOUR, Department of Chemical and Biological Engineering, Montana State University — A generalized short-time expansion of hydrodynamic dispersion is derived using non-linear response theory. The result is in accordance with the well-known reduced cases of shear flow in ducts and pipes. In terms of viscous dominated (low Reynolds number) flow in porous media the generalized expansion facilitates the measurement of permeability by PGSE-NMR measurement of time dependent molecular displacement dynamics. To be more precise, for porous media characterized by a homogeneous permeability coefficient along the direction of flow K, and fluid volume fraction ε, the effective dispersion coefficient $D(t) = \langle|\mathbf{R}\text{-}\langle\mathbf{R}\rangle|^2\rangle/6t$ of molecular displacements $\mathbf{R}$ due to flow and diffusion for a saturating fluid of molecular diffusivity κ in viscous dominated flow is shown to be partially governed by the coefficient of permeability at short times. The short-time expansion is shown to be in agreement with pulsed field gradient spin echo NMR measurement of $D(t)$ in a random sphere pack media and analogous pore-scale random-walk particle tracking transport simulation.

2:21PM R16.00008 Instability of methane hydrate stability zone in permafrost deposits[1] , DMITRIY LYUBIMOV, Perm State University, Perm, Russia, EKATERINA KOLCHANOVA, TATYANA LYUBIMOVA, Institute of Continuous Media Mechanics UB RAS, Perm, Russia, OLEG ZIKANOV, University of Michigan-Dearborn, Dearborn, MI, USA — As a part of a broad study of the dynamics of natural deposits of methane hydrates, we analyze the instability of the interface between the hydrate-bearing zone and the underlying gas-saturated layer. Conditions of Arctic permafrost with temperature below the ice melting point are considered. The physical model includes the Darcy filtration of gas, conduction and convection heat transfer, and the dynamic boundary conditions including the hydrate dissociation at the interface. The method of linear stability analysis is used. It is found that the system is stable and, thus, can exist only at relatively small thicknesses of both layers and at low-to-moderate permeability of the sediments. At larger thicknesses and higher permeability, the interface between the two layers is unstable to monotonic perturbations. The results do not support the hypothesis that the interface instability may lead to accelerating self-sustained dissociation of natural methane hydrates in the conditions of increasing global temperature.

[1]The work is supported by CRDF (grant RUP1-2945-PE-09) and by RFBR (grant 09-01-92505).

2:34PM R16.00009 Direct simulation of the flow over a porous layer of large porosity , PANAGIOTIS-DIMITRIOS ANTONIADIS, MILTIADIS V. PAPALEXANDRIS, Universite Catholique de Louvain — In this talk we report on direct numerical simulations of constant-density flow over and through a layer of a porous medium with large porosity. Initially the fluid is at rest and the flow is driven by a constant pressure gradient. Periodic boundary conditions are used along the streamwise direction, whereas no-slip conditions are specified on the bottom boundary which also coincides with the lower end of the porous strip. Further, outflow conditions are imposed on the top boundary of the computational domain, which is located sufficiently far from the porous medium. As the flow evolves, a boundary layer is formed on the lower end of the porous strip and an additional transition zone is formed right above its upper end. Due to the steep velocity gradients across this zone, a Kelvin-Helmholtz instability is onset which leads to the formation of a mixing layer. We present and analyze the characteristics of vortex pairing and growth rate of this mixing layer. Finally, we discuss the results of a parametric study with respect to the porosity of the medium.

2:47PM R16.00010 Micro-scale flow simulation and colloid transport modeling in saturated porous media[1] , QUEMING QIU, HUI GAO, JIE HAN, YAN JIN, LIAN-PING WANG, University of Delaware — Adequate understanding of the mechanisms governing colloid retention by soil porous media is essential to the prediction and monitoring of the transport of contaminants through groundwater in the subsurface environment. In this talk, we focus on the representation of micro-scale flow and colloid-grain surface interactions in a computational approach. First, micro-scale viscous flows in a model porous media with different domain sizes and geometric configurations are simulated by the mesoscopic lattice Boltzmann method. A Lagrangian colloid tracking model is then used to study the dynamics of colloidal particles under the action of Brownian force, hydrodynamic forces, and physicochemical forces. The modeling and analysis of colloid transport will incorporate the effects of flow speed, solution ionic strength, collector surface roughness, and blocking effect, etc. Simulation results are used to study the unique nature of retention by the secondary energy minimum. Comparisons are made with parallel experimental results obtained from confocal microscopy. To speed up our colloid tracking modeling, parallel implementation using Message Passing Interface (MPI) is performed and the related scalability results will also be presented.

[1]Work supported by USDA and NSF.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R17 Microfluids Mixing 320

12:50PM R17.00001 Measuring Convective and Diffusive Mixing in Inertial Droplet-Pair Collisions , BRIAN CARROLL, CARLOS HIDROVO, The University of Texas at Austin — Complete mixing, dilution, and sample homogenization are essential processes in modern Lab on a Chip and Micro Total Analysis Systems and these seemingly simple tasks remain a major obstacle. A new mixing technique has been proposed that accelerates mixing rates in droplets through controlled, high speed droplet-pair collisions. The collisions take place inside a confined microchannel and the droplet generation and entrainment processes are provided by an inertial gaseous flow. The fast time scales, small length scales, and highly Lagrangian nature of discrete droplet collisions makes optical diagnostics the obvious choice for understanding, characterizing, and quantifying mixing processes. Presented here is a simple and robust visualization and measurement technique that captures convective and diffusive mixing inside droplets using differential fluorescence. High speed digital imagery, custom image processing, and fluorescent intensity statistical analysis are employed to examine the contribution of convective rearrangement and tracer diffusion to droplet mixing following inertial droplet-pair collisions.

1:03PM R17.00002 Mixing in Three Dimensional Linear Spiral Microchannels , MICHAEL FECHTMANN, MATT AMBRUSCH, SHAHAB SHOJAEI-ZADEH, Rutgers University — Since its introduction a decade ago, soft lithography has drastically changed our understanding of fluid physics under confinement by enabling rapid production of microchannels and microdevices. Despite its popularity however, soft lithography has some limitations, including fabrication of thoroughly three dimensional passages without using multiple layers. In this talk, we first introduce the fabrication of a simple three dimensional linear spiral microchannel in a polydimethylsiloxane (PDMS) mold. In two dimensional planar spiral microchannels, Dean number increases or decreases with each consecutive rotation as the radius of curvature decreases or increases, respectively. In our three dimensional spiral microchannel, however, the radius of curvature is constant and as a result the Dean number assumes a constant value. The mixing performance of two miscible fluids in this microchannel is characterized using digital image processing techniques. The influence of flow rate as well as the length of the spiral pitch on the mixing performance is studied. Other possible applications are also discussed.

1:16PM R17.00003 Optimal Blinking-Flow Microfluidic Mixers[1] , DAVID MOTT, Naval Research Laboratory, KEVIN MCILHANY, United States Naval Academy, ELAINE ORAN, Naval Research Laboratory, STEPHEN WIGGINS, University of Bristol — The performance of blinking-flow microfluidic mixers is explored in order to identify optimal mixer designs. A two-dimensional lid-driven flow model is defined that approximates the cross-channel flow in three-dimensional grooved mixers. On either side of a specified separatrix location on the channel floor, the model imposes two different transverse velocities that generate a pair of counter-rotating vorticies of differing sizes. Blinking-flow mixers are then defined by alternating two such fields with the separatrix location for each field chosen independently. An exhaustive search of this design space demonstrates that the best blinking flow mixers are not symmetric, i.e., the second velocity field is not the mirror image of the first. The best mixers combine a field with the separatrix near the channel centerline with a field with separatrix near one of the side walls. Results are compared to comparable three-dimensional grooved channel mixers, and implications of these results for optimizing general mixer designs are discussed.

[1]This work is sponsored by the Office of Naval Research.

1:29PM R17.00004 Use of Eulerian Indicators to Predict Best Mixing Configurations for a Blinking 2D Lid-Driven Flow , K. MCILHANY, U S Naval Academy, D. MOTT, Naval Research Laboratory, S. WIGGINS, University of Bristol, UK, E. ORAN, Naval Research Laboratory — The 2D lid driven model with an alternating flow between two double gyres whose relative size differs is used as the basis of a study to determine the predictive capabilities of two Eulerian Indicators (EI), dubbed the "transversality" and "mobility" with respect to the degree of mixing achieved. The "transversality" EI measures the angular difference between the two alternating velocity vectors at a given position in the flows domain. Experiments have indicated that streamline crossing between two alternating flows is associated with regions of good mixing. The "mobility" EI measures the percentage of the flows domain that contributes the most to particle transport. In the parameter space under study, the product of the two EI's is shown to correlate well with the variance of concentration for the fluid, calculated as a Lagrangian metric. The computational efficiency gained by calculating Eulerian Indicators compared to Lagrangian metrics allows for a more efficient search through this systems parameter space, suggesting configurations which are better suited to mix well, effectively cutting the design time for optimizing new mixing designs.

1:42PM R17.00005 ABSTRACT WITHDRAWN —

1:55PM R17.00006 Fluid Mixing from Viscous Fingering , RUBEN JUANES, BIRENDRA JHA, LUIS CUETO-FELGUEROSO, MIT — Fluid mixing is an important and complex phenomenon. It plays a fundamental role in natural processes, including groundwater flows in heterogeneous media, reactive flows, mantle convection, debris gravity currents, and bacterial locomotion. Mixing at low Reynolds numbers is a notoriously difficult problem because it cannot rely on turbulence. Mixing efficiency at low Reynolds numbers can be enhanced by exploiting hydrodynamic instabilities that induce heterogeneity and disorder in the flow. The unstable displacement of fluids with different viscosities, or viscous fingering, provides a powerful mechanism to increase fluid-fluid interfacial area and enhance mixing. Here we describe the dissipative structure of miscible viscous fingering, and propose a two-equation model for the scalar variance and its dissipation rate. Our analysis predicts the optimum range of viscosity contrasts that, for a given Péclet number, maximizes interfacial area and minimizes mixing time. In the spirit of turbulence modeling, the proposed two-equation model permits upscaling dissipation due to fingering at unresolved scales.

2:08PM R17.00007 Experimental investigation of periodic lines in 3D lid-driven cylindrical cavity flows , JEMIL ZNAIEN, R.R. TRIELING, M.F.M. SPEETJENS, H.J.H. CLERCX, Eindhoven University of Technology, The Netherlands — Mixing of mass and heat by laminar flows occurs in many natural systems and industrial processes. Understanding of the basic mechanisms to enhance mixing efficiency is mostly based on mathematical analysis and numerical studies of prototype cases. Three-dimensional (3D) experiments, however, remain largely unexplored. The present investigation employs 3D Particle Tracking Velocimetry (3D-PTV) to investigate a few of these concepts in realistic experimental configurations. We focus on the observability of periodic lines in periodically lid-driven cylindrical cavity flows. Periodic lines play a central role in the transport properties. The fluid is set in motion via a time-periodic forcing protocol (piece-wise steady translations of one of the endwalls of the cylinder), 3D-PTV measurements have been performed to obtain the web of tracer paths. Experimental results confirm key features from theoretical analysis and numerical studies on the location and shape of the periodic lines. A hybrid method (numerical tracking of particles in an Eulerian flow field determined by experimental measurements) is used to extend the forcing to situations inaccessible by direct particle tracking via 3D-PTV.

2:21PM R17.00008 Reaction-diffusion in microdroplets: Theory & Experiments , ETIENNE FRADET, CHARLES N. BAROUD, LadHyX — We study the dynamics of the reaction front that forms as two initially separated reactants meet. The reactants are initially contained in two different nanoliter droplets confined in a microfluidic device. Guiding and trapping the drops is performed using the *rails* and *anchors* technique. A laser pulse then triggers the chemical reaction by coalescing the drops and we monitor the integral of the reaction product. An asymptotic analysis (Trevelyan, Phys. Rev. E, 80, 2009) identifies two phases for this process. The production rate of the reaction is determined by diffusion in both. Initially the two reactants occupy different regions and have to diffuse to react. The product concentration integral then varies as $t^{3/2}$ with a prefactor that depends on the reaction kinetics. At large times, the reaction rate becomes limited by the diffusive supply of reactants which must travel over longer distances. The product concentration integral then increases as $t^{1/2}$ with a different prefactor containing the same physical ingredients. Confronting theory and experiment allows the measurement of physical and chemical constants.

2:34PM R17.00009 Exploiting numerical diffusion to study transport and chaotic mixing for extremely large Péclet values , PATRICK ANDERSON, MASSIMILANO GIONA[1], OLEKSANDR GORODETSKYI, TU Eindhoven — We show that the purely convective mapping matrix approach provides an extremely versatile tool to study advection-diffusion processes for extremely large Péclet values (∼108 and higher). This is made possible due to the coarse-grained approximation that introduces numerical diffusion, the intensity of which depends in a simple way on grid resolution. This observation permits to address fundamental physical issues associated with chaotic mixing in the presence of diffusion. Specifically, we show that in partially chaotic flows, the dominant decay exponent of the advection diffusion propagator will eventually decay as Pe-1 in the presence of quasiperiodic regions of finite measure, no matter how small they are. Examples of 2d and 3d partially chaotic flows are discussed.

[1]Sabbatical visitor from La Sapienza Università di Roma

2:47PM R17.00010 Visualizing millisecond chaotic mixing in droplets moving through a serpentine microchannel , SHUHUAI YAO, LIGUO JIANG, YAN ZENG, HONGBO ZHOU, JIANAN QU, The Hong Kong University of Science and Technology — We have developed a two-photon excitation fluorescence lifetime imaging technique to accurately and quantitatively measure mixing of two fluorescence dyes inside microdroplets. The line scanning along the microfluidic channel is passively achieved via the droplets flowing through the excitation focal point. Because the periodically generated droplets are identical, we scan multiple droplets and sum up the line signals of each droplet by cross/autocorrelation to obtain the line signal with a high signal-to-noise ratio. The droplets are scanned line by line by moving the focal point across the channel using a translation stage. The cross-sectional image of the droplet is then formed by aligning the scanning lines across the channel. A non-fitting method based on the ratio of fluorescence signals in lifetime decay is used for mixing ratio calibration. With this new imaging technique, we visualize millisecond chaotic mixing dynamics in microdroplets with 5 microsecond time resolution. The mapped chaotic mixing patterns match well with the 2D numerical simulation, performed based on the coupled Laminar two-phase flow level set model and transport of diluted species model, and also validate the characteristics of the alternative asymmetric vortex flow in droplets moving through a serpentine channel.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R18 Acoustics II: Internal and External Flows 321

12:50PM R18.00001 A simple sound source for temporally-developing mixing layers , LIN ZHOU[1], University of Science and Technology of China, MINGJUN WEI, New Mexico State University, DEJUN SUN, University of Science and Technology of China — Applying Lighthill's acoustic analogy equation to temporally-developing mixing layers, we derived a direct relation between the near-field work by pressure fluctuation and the far-field sound. The sound radiation predicted by the new model was compared to the direct numerical simulation, and the results agreed well for the sound radiation from different vortex dynamics in mixing layers: roll-up, pairing (or tearing), merging, saturation, and viscous damping. Using the new formulation, we explained the mechanism for sound generation from the considered flow, and depicted general topological structures for the distribution of sound sources.

[1]Also at: New Mexico State University

1:03PM R18.00002 Large-amplitude acoustic streaming , GREG CHINI, University of New Hampshire — A mechanism for the generation of large amplitude acoustically-driven streaming flows is proposed. Motivated by streaming observed in high-intensity discharge (HID) lamps, two-dimensional flow of an ideal gas in a channel geometry is analyzed in the asymptotic limit of high frequency forcing. Predictions of streaming flow magnitudes based on classical arguments invoking Reynolds stress divergences originating in viscous boundary layers are orders of magnitude too small to account for the observed mean flows. Moreover, classical "Rayleigh streaming" theory cannot account for the direction of the cellular mean flows often observed in HID lamps. In contrast, the inviscid mechanism proposed here, which invokes fluctuating baroclinic torques away from viscous and thermal boundary layers, can account both for the magnitude and the orientation of the observed streaming flows.

1:16PM R18.00003 Control of Acoustic Reflections in a Closed-Circuit Wind Tunnel[1] , MATTHEW KUESTER, EDWARD WHITE, Texas A&M University — Closed return wind tunnels, such as the Klebanoff–Saric Wind Tunnel (KSWT) at Texas A&M University, can provide low-disturbance flows that are required to study boundary layer receptivity. Receptivity is the process through which environmental disturbances become the initial condition for boundary layer instabilities. One instability mechanism, Tollmien–Schlicting waves, is especially receptive to freestream sound. The receptivity of these waves is studied by introducing downstream-traveling planar sound waves that interact with the leading edge of a flat plate; however, changes in wind tunnel area create reflected waves that complicate the experiment. Reflections are mitigated using a secondary speaker located downstream of the test section that eliminates upstream- traveling reflections. The secondary speaker is controlled using a finite impulse response (FIR) filter. Microphone measurements are used to document the wave cancellation at multiple locations in the test section.

[1]The authors gratefully acknowledge the support of NASA and AFOSR through AFOSR grant FA9550-09-1-0341. MSK also acknowledges the support of NASA/ASEE through a NASA Aeronautics Fellowship.

1:29PM R18.00004 Non-Linear High Amplitude Oscillations in Wave-shaped Resonators[1] , DION ANTAO, BAKHTIER FAROUK, Department of Mechanical Engineering and Mechanics, Drexel University — A numerical and experimental study of non-linear, high amplitude standing waves in "wave-shaped" resonators is reported here. These waves are shock-less and can generate peak acoustic overpressures that can exceed the ambient pressure by three/four times its nominal value. A high fidelity compressible axisymmetric computational fluid dynamic model is used to simulate the phenomena in cylindrical and arbitrarily shaped axisymmetric resonators. Working fluids (Helium, Nitrogen and R-134a) at various operating pressures are studied. The experiments are performed in a constant cross-section cylindrical resonator in atmospheric pressure nitrogen and helium to provide model validation. The high amplitude non-linear oscillations demonstrated can be used as a prime mover in a variety of applications including thermoacoustic cryocooling.

[1]The work reported is supported by the US National Science Foundation under grant CBET-0853959.

1:42PM R18.00005 Rijke tube with flexible walls , SHREYAS MANDRE, BAO-NHAT NGUYEN, MARVIN LI, Brown University — Sound is excited spontaneously in a Rijke tube because the small temperature perturbations in an acoustic field interact with heat transfer from a heat source in the tube. The air particles near the heat source undergo a thermodynamic cyle converting heat to mechanical energy, which is heard as the sound emanating from the Rijke tube. This principle of energy conversion is used in thermoacoustic engines, and the main objective of this study is to improve their performance. The acoustic oscillations in the Rijke tube regulate the thermodynamic cycle, just as in conventional engines the cycle is controlled by the motion of a piston and the action of inlet and exit valves. The acoustic regulation in the Rijke tube, however, does not allow arbitrary control of the cycle in thermodynamic phase space. In this presentation, we introduce a new way of overcoming this limitation, one by using Rijke tubes with flexible walls. We will discuss how this modification allows for more general thermodynamic cycles to be executed by the air particles in the tube. This possibility, when used in thermoacoustic engines, opens a channel for further improving the engine performance.

1:55PM R18.00006 Computational study of flow noise from small gaps in turbulent boundary layers[1] , JIN HAO, MINSUK JI, MENG WANG, University of Notre Dame — The noise induced by small gaps underneath low-Mach-number turbulent boundary layers is studied using large-eddy simulation and Lighthill's equation. The latter is solved by employing an acoustically compact Green's function for the gap and by a boundary-element method. The gap leading-edge height is 13% of the boundary-layer thickness, and the gap width and trailing-edge height are varied to investigate their effect on sound generation. The radiated acoustic field is dominated by the forward-facing step in the gap and resembles forward-step noise for wide gaps and/or asymmetric gaps with the trailing edge higher than the leading edge. For narrow and symmetric gaps, destructive interference of the sound from leading and trailing edges causes a significant decline in the low-frequency spectral content and thereby creates a broad spectral peak in the mid-frequency range. The effect of acoustic noncompactness of gaps is investigated by comparing solutions based on a compact Green's function and those from a boundary-element calculation. Excellent agreement is observed at low frequencies and away from the wall-normal direction. At higher frequencies, the sound field deviates from that of a compact streamwise dipole. The elevated level of surface pressure fluctuations induced by gaps and their recovery to equilibrium conditions are also examined.

[1]Supported by ONR Grant N00014-09-1-0602

2:08PM R18.00007 Investigation of a tone in flap tip noise , RAFAEL S. GIORIA, STERGIOS P. TSILOUFAS, NDF, Dept. Mech. Eng., Escola Politecnica, University of Sao Paulo, FRANCISCO K. KLUG, Embraer, JULIO R. MENEGHINI, NDF, Dept. Mech. Eng., Escola Politecnica, University of Sao Paulo — In this paper, we investigate numerically the noise generated by the flow around a high-lift configuration. The case studied is the noise generated by the flap tip of the MDA-30P30N high-lift geometry. We propose a cavity model as a simplification for the flap cove flow near the flap tip. The idea of representing the flap cove as a cavity is due to a tone in the spectrum from the numerical simulations of the full high lift geometry MDA-30P30N without spanwise gap between extended flap and the stowed flap. A high peak around 900Hz to 1000Hz was not expected and its source is not clear. The effort here is to investigate the source of frequencies around this value. There is a possibility of noise tones generated from the flap cove with frequencies around 900Hz according to the Rossiter model for the cavity. This mechanism of noise generation seems to be associated to the peak in the spectrum. Snapshot of instantaneous vorticity field near the flap tip supports that a mechanism of noise generation resembles the cavity mechanism. Further corroboration is presented through the Koopman modes (dynamic mode decomposition) of the pressure fields.

2:21PM R18.00008 Unsteady Lift and Radiated Sound Generated by a 2-D Airfoil in an Intermittent Flow , MARK ROSS, SCOTT MORRIS, University of Notre Dame — The spanwise correlation length scale of lateral velocity and the gust response function are the quantities of interest in predicting the sound production from an airfoil. Typically, these quantities are taken to be a correlation length scale model based on isotropic turbulence and Sears' gust response function, respectively. The present study is an experimental investigation of the accuracy of these selections when the turbulent approach flow is intermittently irrotational. Acoustic measurements of a flat-plate airfoil placed at three lateral locations in a single-stream shear layer are presented. The acoustic measurements are compared to radiated sound predictions based on detailed velocity field measurements.

2:34PM R18.00009 Investigation of noise production from a turbulent cylinder , ZANE NITZKORSKI, KRISHNAN MAHESH, University of Minnesota — We investigate noise production by a cylinder in turbulent flow (Re=10,000 and M=0.2), using the Ffowcs-Williams and Hawkings acoustic analogy, where the sound sources are computed from a compressible direct numerical simulation. It has been shown that vortices passing through a data surface generate spurious noise if the quadrupole term is neglected. Our implementation of the acoustic analogy, hence, uses porous data surfaces as well as the volume term. We compare our solution to available results, examine the effect of the placement of the data surface on the noise calculation by using multiple surfaces, and the effect of different proposed correction schemes that try to compensate for neglecting the quadrupole noise.

2:47PM R18.00010 Acoustic Scattering from Interaction of Dual Frequency Incident Fields , MAX DENIS, CHRISNA NGUON, KAVITHA CHANDRA, CHARLES THOMPSON, University of Massachusetts Lowell — The pressure field produced by the spatial interaction of two high frequency incident plane waves in a three-dimensional scattering object is investigated. Of particular interest is the field produced in response to the difference-frequency component generated from the non-linear interaction of the two harmonically time varying acoustic beams in a high contrast medium. The scattered pressure at the difference frequency field has been shown to enhance the identification of abnormal biological tissue in recent medical ultrasound experiments. This work presents a computational study of the scattered pressure that results from the Reynolds stress in a fluid scatterer. Using Pade approximants, it is shown that the stress tensor can be computed using a uniform expansion in the contrast gauge for the scattered pressure. This allows one to investigate scattering volumes characterized by high compressibility contrast.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R19 Instability General II 322

12:50PM R19.00001 Liquid-Bridge Breaking Limits , ASHLEY MACNER, PAUL STEEN, Cornell University — Wet adhesion by liquid bridges in large arrays shows promise for use in lightweight, controllable on-demand devices. Applications include grab/release of wafer substrates, transport of micron-sized tiles for use in 3D printing and micro-dosing of personalized pharmaceutical drugs. By wetting and spreading, a drop can form a bridge and thereby "grab" a nearby solid substrate. By volume decrease or extension, the bridge can break. The breaking limit corresponds to bridge instability which can be predicted, knowing the static mechanical response of the bridge. Mechanical behaviors include force-volume (FV), pressure-volume (pV) and force-length (FL) responses. Instability crucially depends on the mode of failure – failure under constant-force or constant length are typical cases. We study single bridge equilibria for their breaking limits. FV diagrams for the pin-pin equal and pin-pin unequal radii boundary conditions for different bridge heights are measured in the laboratory. The FL response in the case of pin-pin equal radii is also measured. Results are compared to predictions of static theory. Static results are then used to compare to dynamical sequences where volume is driven quasistatically by syringe or an electro-osmotic pump. As the breaking limit is approached, the shape deformation accelerates leading to non-equilibrium shapes not captured by the static analysis.

1:03PM R19.00002 Scenarios on a rotating two-fluid interface with a density contrast - the morphology and the transitions[1] , WEI-TING LIN, CHING-YAU LAI, Dept. of Physics, Nat'l Taiwan University, CHI-CHUNG CHANG, Inst. of Physics, Academia Sinica, YIH-YUH CHEN, Dept. of Physics, Nat'l Taiwan University, JIH-CHIANG TSAI, Inst. of Physics, Academia Sinica, INST. OF PHYSICS, ACADEMIA SINICA COLLABORATION, DEPT. OF PHYSICS, NAT'L TAIWAN UNIVERSITY COLLABORATION — We study experimentally an oil-water interface maintained in a cylindrical container with its upper boundary rotating at constant speeds. The interface exhibits intriguing morphology as the rotation speeds up, making transitions from a smooth hump that presumably compensates the centrifugal-force induced pressure dip, to more fascinating geometries such as a spinning flap top (a plateau) or a mound with distinct spatial steps. The available scenarios can be controlled by varying the depths of two fluids as well. Increasing the rotation rates also tends to induce wavy patterns that break the axial symmetry of those base shapes, while a violent collapse of the oil-water interface occurs at sufficiently fast rotations. We attempt to give partial explanations for the different scenarios.

[1]Supported by Academia Sinica and NSC, Taiwan.

1:16PM R19.00003 Role of intrinsic flame instability in the excitation of combustion chamber instability , V'YACHESLAV AKKERMAN, CHUNG K. LAW, Princeton University — While considerable progress was made on understanding the various modes of flame instability at the fundamental level, and substantial empirical information and phenomenological descriptions was also accumulated on combustion instability within combustion chambers such as those of rocket engines, few attempts were made to explore the possible macro-scale excitation of the latter through the micro-scale manifestation of the former. Here we present an initial attempt towards identifying such a possibility and the associated coupling mechanisms. We shall incorporate the flame parameters into the classical theories of liquid-propellant rocket engines, and then implement the rocket dynamics into the analyses of premixed and diffusion flame segments. The analyses are conducted for the various instability modes, including the diffusional-thermal, Darrieus-Landau, and Rayleigh-Taylor (body-force) instabilities for premixed flames, and the Kelvin-Helmholtz and body-force instabilities for diffusion flames. The role of chamber-generated sound on stabilizing the inherent flame instabilities and triggering the parametric instability is also considered.

1:29PM R19.00004 Experiments on the Mode Selection in Faraday Instability[1] , WILLIAM BATSON, Univ Florida, USA, FARZAM ZOUESHTIAGH, IEMN, Univ. Lille, France, RANGA NARAYANAN, Univ Florida, USA, UNIV FLORIDA, USA TEAM, IEMN, UNIV. LILLE, FRANCE TEAM — The resonance of a fluid interface with an oscillating acceleration field is studied for liquid-liquid systems in small aspect ratio cylindrical and rectangular containers. The resonant phenomenon, known as Faraday waves, is more typically studied in large aspect ratio systems at high frequencies where multiple modes are excited simultaneously and the associated nonlinear interactions give way to a variety of patterns. In this work the low excitation frequency allows for the selection of individual cell modes and their dynamics are considered. The onset threshold for instability is compared to the predictions of this model. It is seen for the cell modes that frequency bands are well predicted by the model and the amplitudes are very close, with deviation attributable to interfacial pinning and sidewall stresses. Supercritical and subcritical bifurctions are observed, along with other nonlinear phenomena such as co-dimension2 points and wave breaking.

[1]Support from NSF OISE 0968313, Partner Univ. Fund and a Chateaubriand Fellowship is acknowledged.

1:42PM R19.00005 Spatially localized patterns in 2D and 3D doubly diffusive convection , CEDRIC BEAUME, ALAIN BERGEON, IMFT (France), EDGAR KNOBLOCH, Department of Physics, UC Berkeley, IMFT (FRANCE) TEAM, DEPARTMENT OF PHYSICS, UC BERKELEY TEAM — Doubly diffusive convection, that is, convection driven by a combination of concentration and temperature gradients, is known to display a wealth of dynamical behavior whose properties depend on the gradients. In the present work, we first investigate spatially localized states in two-dimensional horizontal thermosolutal convection with no-slip boundary conditions at top and bottom and vertical gradients of temperature and concentration. Numerical continuation demonstrates the formation of stationary convectons in the form of 1-pulse and 2-pulse states of both odd and even parity while time integration reveals the presence of stable time dependent spatially localized states. We next turn to large scale three-dimensional vertical enclosures placed in horizontal thermal and solutal gradients. Different types of spatially localized states are computed and the results related to the presence of homoclinic snaking.

1:55PM R19.00006 Modulation instability of space-periodic oscillatory patterns , ALEXANDER NEPOMNYASHCHY, Technion – Israel Institute of Technology, SERGEY SHKLYAEV, University of Puerto Rico, ALEXANDER ORON, Technion – Israel Institute of Technology — Pattern selection and stability of regular (periodic in space) regimes is a classical problem with a number of applications in fluid dynamics. For steady bifurcations both competition of perfect periodic patterns and their stability with respect to slow modulations in space (e.g. Eckhaus or zigzag instabilities) are well studied. In contrast, in the case of Hopf bifurcation, usually only selection of patterns that possess a certain symmetry was analyzed (Silber & Knobloch, Nonlinearity, 1991; Roberts et al, Contemp. Math, 1986), whereas the set of Ginzburg-Landau equations was studied only in the one-dimensional case (rolls). Dealing with a wide class of problems, where the longwave oscillatory instability takes place, we consider a stability of regular oscillatory patterns that belong to either square or hexagonal lattices with respect to spatial modulations. By means of the multiple scale expansion, we derive instability criteria valid near the stability threshold. Useful classification of possible perturbations of a regular structure is introduced. As an example, the theory is applied to Marangoni convection in a layer of a binary mixture with the Soret effect. Domains of stability of space-periodic patterns are obtained.

2:08PM R19.00007 ABSTRACT MOVED TO E20.00007 —

2:21PM R19.00008 Break of the symmetry in a two-lid driven cavity[1] , THOMAS LEMEE, Univ. Paris-Sud, Laboratory F.A.S.T, France, Orsay, 91401, GERARD LABROSSE, Univ. Paris-Sud, Department of Physic, France, Orsay, 91401, GUILLAUME KASPERSKI, RANGA NARAYANAN, Univ. Florida,Department of Chemical Eng., Gainesville FL 32611, UNIV. PARIS-SUD, LABORATORY F.A.S.T TEAM, UNIV. PARIS-SUD, DEPARTMENT OF PHYSIC TEAM, UNIV. FLORIDA, DEPARTMENT OF CHEMICAL ENG. TEAM — The lid driven cavity has applications in crystal growth as well as in the coating industry. We study the problem of a driven cavity with two parallel walls moving at the same speed and in the same direction. Time marching calculations using Chebyshev-spectral method were done with different aspect ratios. As the Reynolds number increases, the onset of the instability is characterized by the break of the symmetry which is described and explained. The critical Reynolds number depends on the aspect ratio. This dependence is explained.

[1]Support from NSF OISE 0968313, the Partner University Fund.

2:34PM R19.00009 Transition to turbulence in a quasi-2D Kolmogorov flow , JEFFREY TITHOF, BALACHANDRA SURI, RADFORD MITCHELL, A.J. PRYOR, ROMAN GRIGORIEV, MICHAEL SCHATZ, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology — We describe a combined experimental and numerical study of quasi-2D flows to search for unstable, exact solutions to Navier-Stokes known as Exact Coherent Structures (ECS), which may provide a foundation for a simplified dynamical description of turbulence. We focus on a system that closely approximates Kolmogorov flow by inducing shear in a thin fluid layer using electromagnetic forces. PIV is used to obtain time series of velocity fields from images of the visualized flows in the lab; time series of velocity fields are calculated numerically for flows with forcing similar to that in the experiments. Discrepancies arising from differences in lateral boundary conditions between experiments (no slip) and simulations (periodic) are addressed in two separate ways: (1) experimentally studying a large system to approximate the effects of periodic boundary conditions and (2) adding padding regions in the simulations to mimic finite system size. We describe in detail the sequences of bifurcation leading to turbulence in both experiments and simulations.

2:47PM R19.00010 ABSTRACT WITHDRAWN —

Tuesday, November 22, 2011 12:50PM - 2:47PM –

Session R21 Vortex Dynamics VII 324-325

12:50PM R21.00001 How leakage flow influences the hydrodynamic damping of a vibrating blade[1] , MATTHIEU DREYER, ETIENNE CARTIER, STEVEN ROTH, MOHAMED FARHAT, Ecole Polytechnique Federale de Lausanne — The hydrodynamic damping of a structure is of particular importance in many engineering applications. In the case of axial turbines, the presence of a gap between the rotor tip and the shroud induces a leakage flow creating a tip vortex whose roll up process is highly dependent on the clearance size. The blade response to an excitation in the presence of this flow is however poorly characterized. In the present study, the hydrodynamic damping of a Naca hydrofoil in a water tunnel is investigated with respect to the gap width. An innovative device was designed to excite the hydrofoil in non-intrusive way: an underwater electric discharge creates a fast growing and collapsing bubble which generates strong shockwaves. The structural response is monitored with a Laser Vibrometer. Assuming a single degree of freedom system, the hydrodynamic damping for the first two eigen modes (bending and torsion) is identified for different values of upstream velocity, incidence angle and tip clearance size.

[1]Competence Center for Energy & Mobility AND Swisselectric Research

1:03PM R21.00002 Counter rotating open rotor flow field investigation using stereoscopic Particle Image Velocimetry , ERIC ROOSENBOOM, ANDREAS SCHROEDER, REINHARD GEISLER, DIETER PALLEK, JANOS AGOCS, DLR - German Aerospace Center, KLAUS-PETER NEITZKE, Airbus Operations GmbH — Counter rotating open rotor (CROR) propulsive systems are again considered as fuel efficient alternatives to conventional propulsion systems. In the present paper details of dedicated experiments on a generic CROR model are studied using stereoscopic Particle Image Velocimetry. The CROR model has 10 front blades and 8 aft blades, with blade design similar to modern propellers for high disk loadings. Recent progress in Particle Image Velocimetry applications to propeller flow analysis is used to further develop the technique for application to CROR systems. Stereoscopic Particle Image Velocimetry (SPIV) has been applied for the flow field investigation behind a counter rotating open rotor (CROR) model in order to enable experimental insight in the complex flow phenomena of multiple vortex structures. The paper discusses a dedicated triggering strategy for the determination of the phase positions of both propellers using the phase delays and revolution periods. Results of the PIV measurements are presented and the topological events of the rotor-interactions are discussed.

1:16PM R21.00003 A comparison between the growth of leading-edge and tip vortices on a low aspect ratio plate using 3D-PTV , COLIN HARTLOPER, DAVID RIVAL, DAVID WOOD, University of Calgary, MATTHIAS KINZEL, California Institute of Technology — The vortex formation process on an impulsively started flat plate at 45 degree incidence is studied using both direct force measurements as well as 3D-PTV. The focus of this experiment has been to study the competing evolution of leading-edge and tip vortices for a range of start-up motions. The 3D-PTV measurement volume, located in the tip region of the plate, provides detailed insight into the roll-up of the shear layers at early stages of the motion. By quantifying the evolution of circulation at various spanwise and tip locations, we are able to relate the unsteady generation of force to the instantaneous vortex topology present around the plate.

1:29PM R21.00004 Influence of outlet geometry on the swirling flow in a simplified model of a large two-stroke marine diesel engine , S. HAIDER, T. SCHNIPPER, K.E. MEYER, J.H. WALTHER, Department of Mechanical Engineering, Technical University of Denmark, S. MAYER, MAN Diesel & Turbo, Denmark — We present Stereoscopic particle image velocimetry measurements of the effect of a dummy-valve on the in-cylinder swirling flow in a simplified scale model of a large two-stroke marine diesel engine cylinder using air at room temperature and pressure as the working fluid and Reynolds number 19500. The static model has stroke-to-bore ratio of 4, is rotationally symmetric and the in-cylinder swirling flow is enforced by angled ports at the inlet. We consider a case analogous to engine when the piston is at bottom-dead-center. In absence of an exhaust valve the overall axial velocity profile is wake-like and flow reversal is observed on the cylinder axis, close to the inlet. Downstream, the flow reversal disappears and instead a localized jet develops. The corresponding tangential velocity profiles show a concentrated vortex with decreasing width along the downstream direction. By placing a concentric dummy-valve at the cylinder outlet, the magnitude of reverse flow at the inlet increases, the strong swirl is diminished and the axial jet disappears. We compare these findings with previous measurements in vortex chambers and discuss the relevance of these results with respect to development of marine engines.

1:42PM R21.00005 The effect of non-zero radial velocity on the impulse and circulation of starting jets , MICHAEL KRIEG, KAMRAN MOHSENI, University of Florida — Vortex ring formation dynamics are generally studied using two basic types of vortex generators. Piston cylinder vortex generators eject fluid through a long tube which ensures a purely axial jet; whereas, vortex ring generators which expel fluid through a flat plate with a circular orifice produce 2-D jets (non-zero radial velocity). At the nozzle exit plane of the orifice type vortex generator the radial component of velocity is linearly proportional to the radial distance from the axis of symmetry, reaching a maximum at the edge of the orifice with a magnitude around 10 % of the piston velocity (the ratio of the volume flux and the nozzle area). As the jet advances downstream the radial velocity quickly dissipates, and becomes purely axial less than a diameter away from the nozzle exit plane. The radial velocity gradient in the axial direction plays a key role in the rate at which circulation and impulse are ejected from the vortex generator. Though the radial component of velocity is small compared to the axial velocity, it has a significant effect on both the circulation and impulse of the starting jet because of this gradient. The extent of circulation and impulse enhancement is investigated through experimental DPIV data showing that the orifice device produces nearly double both circulation and energy (with identical piston velocity and stroke ratios).

1:55PM R21.00006 Effects of ground plane topology on vortex-ground interactions in a forced impinging jet[1] , JAYSON GEISER, KYLE CORFMAN, KEN KIGER, University of Maryland — The phenomenon of a three-dimensionally unstable vortex-ground interaction is studied, motivated by the problem of sediment suspension by vortex-wall interactions from landing rotorcraft. In the current work, the downwash of a rotorcraft is simplified using a prototype flow consisting of an acoustically forced impinging jet. The goal of the current investigation is to quantify the effects of disturbances to the ground-plane boundary layer on the three-dimensional development of the vortex ring as it interacts with the ground plane. A small radial fence is employed to perturb the natural evolution of the secondary vortex, which typically exhibits azimuthal instabilities as it is wrapped around the primary vortex. The fence is observed to localize and intensify the azimuthal development, dramatically altering the mean flow in this region and generating corresponding azimuthal variations in the turbulent near-wall stresses. Multi-plane ensemble-averaged stereo PIV is employed to obtain volumetric, phase averaged data sets that are subjected to a triple decomposition to fully quantify turbulence effects. The effects of the radial fence are examined at both a high and low Reynolds number flows ($\mathrm{Re} = \Gamma/\nu = 50{,}000$ and 10,000, respectively), and the data is analyzed in the context of structures leading to significant sediment mobilization.

[1]Current work is funded by the AFOSR under grant FA9550-08-1-0406

2:08PM R21.00007 Mach Number Effect on the Characteristics of a Free Pulsed Jet , ISAAC CHOUTAPALLI, The University of Texas - Pan American — An experimental study was carried out on a free pulsed jet over a Mach number range of 0.3 to 0.8. The data was obtained using Particle Image Velocimetry (PIV). The global flow field of the pulsed jet showed that over the range of Mach numbers considered, the centerline velocity decay, jet spreading and the normalized mass flow rate for a given Strouhal number (St = fd/Uj; where f is the frequency of pulsation, d is the nozzle exit diameter and Uj is the time-averaged axial velocity at the nozzle exit) are only weakly dependent on the nozzle exit Mach number. The phase-averaged results show that the vortex ring circulation is also weakly dependent on the nozzle exit Mach number for a given Strouhal number. The vortex ring pinch-off was found to be independent of the nozzle exit Mach number but dependent on the pulsing frequency.

2:21PM R21.00008 High-speed schlieren videography of vortex-ring impact on a wall , BENJAMIN KISSNER, MICHAEL HARGATHER, GARY SETTLES, Pennsylvania State University — Ring vortices of approximately 20 cm diameter are generated through the use of an Airzooka toy. To make the vortex visible, it is seeded with difluoroethane gas, producing a refractive-index difference with the air. A 1-meter-diameter, single-mirror, double-pass schlieren system is used to visualize the ring-vortex motion, and also to provide the wall with which the vortex collides. High-speed imaging is provided by a Photron SA-1 digital video camera. The Airzooka is fired toward the mirror almost along the optical axis of the schlieren system, so that the view of the vortex-mirror collision is normal to the path of vortex motion. Vortex-wall interactions similar to those first observed by Walker et al. (JFM 181, 1987) are recorded at high speed. The presentation will consist of a screening and discussion of these video results.

2:34PM R21.00009 ABSTRACT WITHDRAWN —

Tuesday, November 22, 2011 12:50PM - 3:00PM —
Session R23 Flow Control VI 326

12:50PM R23.00001 Advanced Fluid Research On Drag reduction In Turbulence Experiments – AFRODITE[1] , JENS H.M. FRANSSON, Linne Flow Centre, KTH Mechanics — A hot topic in today's debate on global warming is drag reduction in aeronautics. The most beneficial concept for drag reduction is to maintain the major portion of the airfoil laminar. Estimations show that the potential drag reduction can be as much as 15%, which would give a significant reduction of NOx and CO emissions in the atmosphere considering that the number of aircraft take offs, only in the EU, is over 19 million per year. In previous tuned wind tunnel measurements it has been shown that roughness elements can be used to sensibly delay transition to turbulence[2]. The result is revolutionary, since the common belief has been that surface roughness causes earlier transition and in turn increases the drag, and is a proof of concept of the passive control method per se. The beauty with a passive control technique is that no external energy has to be added to the flow system in order to perform the control, instead one uses the existing energy in the flow. Within the research programme AFRODITE, funded by ERC, we will take this passive control method to the next level by making it twofold, more persistent and more robust.

[1]Financial support from the European Research Council (ERC) is acknowledged.

[2]Fransson et al. 2006 *Phys. Rev. Lett.* **96**, 064501.

1:03PM R23.00002 Closed-Loop Aerodynamic Flow Control of a Maneuvering Airfoil , DANIEL P. BRZOZOWSKI, JOHN R. CULP, ARI GLEZER, Georgia Institute of Technology — The unsteady interaction between trailing edge aerodynamic flow control and airfoil motion in pitch and plunge is investigated in wind tunnel experiments using a 2-DOF traverse which enables application of time-dependent external torque and forces by servo motors. The global aerodynamic forces and moments are regulated by controlling vorticity generation and accumulation near the surface using 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 effect of the unsteady motion on the model-embedded flow control is assessed in unsteady several maneuvers. Circulation time history that is estimated from a PIV wake survey shows that the entire flow over the airfoil readjusts within about 1.5 TCONV, which is about two orders of magnitude shorter than the characteristic time associated with the controlled maneuver of the wind tunnel model. This illustrates that flow-control actuation can be typically effected on time scales that are commensurate with the flow's convective time scale, and that the maneuver response is primarily limited by the inertia of the platform.

1:16PM R23.00003 A Method to Control Compliance of Blades and Flaps[1] , JULIA COSSE, MORTEZA GHARIB, California Institute of Technology — Compliant plates experience lower drag forces than rigid plates primarily due to reconfiguration. From this concept it follows that through modifying the compliance of a plate, the aerodynamic forces can be controlled. To achieve this control, the concept of hydroskeletons – which are fluid filled cavities that resist deformation through the pressure of an internal fluid – were used. Using this notion a compliant structure with an internal chamber was developed. Shape change was detected when filling the chamber with fluid and controlling the pressure. Preliminary testing involved simple internal geometries filled with water and pressurized up to 20 psi. Using this method a plate was built with several internal chambers, each with individual pressure control. The plate was attached to a force balance perpendicularly in a wind tunnel. Drag and lift forces were modified through changing the internal pressure both globally and locally.

[1]This research is supported by the Gordon and Betty Moore foundation.

1:29PM R23.00004 Manipulation of Laminar Separation Utilizing Dynamic Roughness at the Leading Edge of an Idealized Airfoil , RYAN WALLACE, BEVERLEY MCKEON, California Institute of Technology — Low Reynolds number flow over a symmetric, idealized airfoil with a reasonably constant laminar separation point was manipulated using a leading edge roughness element with small, time-dependent amplitude. At a fixed height and low Reynolds number the roughness element was able to reduce the extent of laminar separation over the airfoil as compared to a smooth airfoil. Further reduction of the separation was achieved by dynamically oscillating the roughness element in an appropriate range of actuation frequencies. Proper orthogonal decomposition performed upon the flow over the airfoil for both the baseline and active open loop case shows the introduction of persistent structures within the flow due to the oscillating roughness element. The coupling of this small input perturbation with the flow and the resultant manipulation of the separation bubble will be discussed for a range of flow and roughness conditions. The support of NSF CAREER award #0747672 is gratefully acknowledged.

1:42PM R23.00005 Vortex shedding response of streamwise driven cylinders , DANIEL TUDBALL SMITH, JUSTIN LEONTINI, JOHN SHERIDAN, Monash University, Australia, DAVID LO JACONO, INPT, UPS, CNRS, IMFT, Toulouse, France — Bluff body cylinders exhibit a range of responses when externally forced. This study experimentally investigated the modes of response and trends in shedding frequency of square and circular cylinders undergoing inline forced oscillations in a steady flow. Experiments were conducted in a free surface water channel over a range of Reynolds numbers from 1500 to 6300. With the driving frequency held constant at the natural unperturbed shedding frequency, the forcing amplitude was varied and the response examined. Frequency analysis of velocity, lift and drag forces showed that for low amplitude oscillation the shedding frequency decreases as amplitude increases, displaying a quadratic relationship. As amplitude increases further a region of possible mode competition exists until a critical amplitude is reached where the shedding locks to a period-doubled subharmonic of the forcing frequency. The region of mode competition is brief with an abrupt lock for the circular cylinder, while the square cylinder exhibits a gradual transition to the subharmonic over a larger range of amplitudes.

1:55PM R23.00006 Transition delay by means of base flow modulations[1] , S.S. SATTARZADEH, S. SHAHINFAR, B.E. FALLENIUS, J.H.M. FRANSSON, A. TALAMELLI, Linne Flow Centre, KTH Mechanics — Recent experimental investigations have shown that spanwise modulations of the base flow may delay transition to turbulence.[2] In this study we explore the possibility to generate streaks of much larger amplitude than previously reported by using a row of miniature vortex generators (MVGs). Here, we present the first boundary layer experiment where streak amplitudes exceeding 30% have been produced without having any secondary instability acting on them. Furthermore, the induced skin-friction drag due to the streaky base flow is quantified and it is demonstrated that the streaks can be reinforced by placing a second array of MVGs downstream of the first one. In this way it is possible to make the control more persistent in the downstream direction. We conclude that the specially designed set of MVGs, as a boundary layer modulator, is a promising candidate for successfully setting up robust and persistent streamwise streaks, which is a prerequisite for a successful flow control. This work is carried out within the AFRODITE programme funded by ERC.

[1]Financial support from the European Research Council (ERC) is acknowledged.

[2]Fransson et al. 2006 *Phys. Rev. Lett.* **96**, 064501.

2:08PM R23.00007 Electroactive Polymer based flow control at Low Reynolds Numbers , SARAH ZAREMSKI, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Electroactive polymers (EAPs) are used to achieve distributed, conformal actuation on aerodynamic surfaces, promote transition in places where laminar separation occurs, and maintain laminar flow where there is no separation. In this work, the feasibility of the EAPs in mitigating a laminar separation bubble on a flat plate was examined. First, the performance of the EAPs was evaluated using a Laser vibrometer and a high-speed camera to quantify the response of the EAP to input voltage and driving frequency and to better understand the physics of the actuator itself. Then, measurements using Stereoscopic particle image velocimetry were conducted in the vicinity of the EAP. Several parameters, such as the driving frequency, the input voltage, and the dimple size, were tested. Preliminary data show promising results, where the size of the separation bubble was significantly reduced when the EAPs were driven at the appropriate voltages and frequencies.

2:21PM R23.00008 Experimental Evaluation of Control Algorithms for a Supercavitating Vehicle , DAVID ESCOBAR SANABRIA, U niversity of Minnesota, ROGER ARNDT, GARY BALAS, University of Minnesota — High speed supercavitating vehicles offer significant challenges regarding control. Vehicles with actuated control surfaces, such as cavitators and fins, are of considerable interest for maneuverability and control. To study the interaction of control surfaces and the body dynamics of the vehicle, a new hardware and software infrastructure has been developed at Saint Anthony Falls Laboratory (SAFL-U of Minn). In addition, a new vehicle prototype that utilizes a cavitator disk and fins for control, a 6 degree-of-freedom force balance to measure forces and moments, and a ventilation system to insure a fully developed supercavity was designed and tested in the high-speed water tunnel at SAFL. Based on experiments in presence of a supercavity surrounding the vehicle, mathematical models that map cavitator and fins angles to pitch moment, drag force and lift force are obtained. These mathematical models and the new platform enable the use of closed-loop control to significantly reduce pitch moment oscillations induced by a gust flow. This achievement shows a promising path towards the experimental validation of control algorithms for high-speed supercavitating vehicles. The platform architecture, experimental design, mathematical models, and validation process are presented here.

2:34PM R23.00009 Bubble Gate for In-Plane Flow Control , ALI KAZEMI OSKOOEI, AXEL GUENTHER, University of Toronto — The ability to control fluid flow is of key importance for microfluidic devices. While a large number of sophisticated solutions have been demonstrated, there is still a great amount of interest in developing simple strategies that do not require complex fabrication steps and electrical connections. A small footprint, compatibility with different substrate materials, working fluids and temperatures are amongst other desirable characteristics. We demonstrate a bubble gate strategy that meets all the above. In this strategy, flow control is achieved using a controlled gas stream that intercepts a liquid stream at a T-junction, forming a gas-liquid interface (i.e. bubble). Closely positioned micropillars are employed to limit the bubble motion to a single degree of freedom. The bubble breaks into the liquid stream and occupies the entire liquid cross-section, when the gas pressure is continued. Hence, the bubble movement is able to stop or manipulate the liquid flow. Several control operations are discussed herein, including, but not limited to, valves, liquid metering and peristaltic pumping. PIV measurements are employed to investigate the transient flow structure.

2:47PM R23.00010 Vertical Fence Wake Manipulation Using Periodic Variation of Upstream Flow , SEUNG-HWAN LEE, XIN CHENG TU, HYOUNG-WOO KIM, HYOUNG-BUM KIM, School of Mechanical Engineering, Gyeongsang National University, Korea — The effect of periodic variation of upstream flow on the separated shear flows behind the vertical fence was experimentally investigated. Upstream flow was modified using small obstacles and this device made the periodic change of streamwise velocities in front of the fence. The experiments were performed in a circulating water channel. The Reynolds number based on the height of fence and freestream velocity was varied from 2000 to 6000. The vertical fence was submerged in the turbulent boundary layer. Stereoscopic-PIV method was used to measure the instantaneous velocity fields around a vertical fence. 800 instantaneous velocity fields were acquired in each experimental condition and the mean properties were calculated using the ensemble average method. The obtained results were compared with those of uncontrolled fence flow. The results showed the vertical fence under the upstream flow change has the local downwash flow behind the fence and this flow suppressed the separation bubble and made the smaller recirculation region compared with uncontrolled fence wake.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R25 Instability in Jets and Wakes II 328

12:50PM R25.00001 Mixing in Long Cylinder by a Stratified Jet: Laboratory Modeling and Theory[1] , C. NATH, S.I. VOROPAYEV, H.J.S. FERNANDO, University of Notre Dame — The evolution of buoyant turbulent jets released into a low aspect ratio (width/height) cavity filled with a homogeneous fluid was investigated experimentally. The motivation was to understand mixing process in U.S. Strategic Petroleum Reserves (SPR), where crude oil is stored in salt caverns of aspect ratio approximately 0.1. During maintenance, degassed oil is introduced as a jet from the top of the caverns while denser gas-laden crude oil is pumped out from the bottom. The focus was on mixing, formation and development of density layer as well as the time for replenishing oil in the container to an acceptable level of vapor pressure (gas concentration). Basing on the results of experiments a theoretical model was advanced which permits to calculate the vertical density distributions in cavern as a function of time and other external parameters. Satisfactory agreement between theory and measurements was demonstrated. The results obtained could be extrapolated to SPR flow mixing situations and help to improve the efficiency of expensive oil cavern refilling.

[1]This research is supported by the Sandia National Laboratories.

1:03PM R25.00002 The visualization of the acoustic feedback loop in impinging underexpanded supersonic jet flows using ultra-high frame rate Schlieren[1] , DANIEL MITCHELL, DAMON HONNERY, JULIO SORIA, Monash University, VIC 3800 Australia — An acoustic feedback model for supersonic jet impingement has been proposed in past literature. Due to the inherent difficulty in measuring highly transient phenomena, models of the feedback process have mostly relied on inference, and comparison to similar subsonic flows. Through the use of ultra-high speed cameras operating at one million frames per second, it has been possible to directly visualize the acoustic feedback loop for the first time. Time resolved Schlieren and shadowgraph image sequences capture the interaction of upstream travelling acoustic waves with the shear layer at the nozzle lip and shock structures within the jet core. The acoustic forcing at the nozzle lip produces a sinusoid like perturbation in the shear layer that is highly transient both temporally and spatially. This perturbation grows rapidly into a Kelvin-Helmholtz like vortex ring. These time resolved measurements offer new insights into the fundamental physical mechanisms governing the acoustic feedback loop in supersonic jet impingement.

[1]The authors acknowledge the support of the ARC Discovery and LIEF grants.

1:16PM R25.00003 The Effect of Orifice Eccentricity on Instability of Liquid Jets , GHOBAD AMINI[1], ALI DOLATABADI[2], Concordia University — The hydrodynamic instability of inviscid jets issuing from elliptic orifices is studied. A linear stability analysis is presented for liquid jets that includes the effect of the surrounding gas and an explicit dispersion equation is derived for waves on an infinite uniform jet column. Elliptic configuration has two extreme cases; round jet when ratio of minor to major axis is unity and plane sheet when this ratio approaches zero. Dispersion equation of elliptic jet is approximated for large and small aspect ratios considering asymptotic of the dispersion equation. In case of aspect ratio equal to one, the dispersion equation is analogous to one of the circular jets derived by Yang [1]. In case of aspect ratio approaches zero, the behavior of waves is qualitatively similar to that of long waves on a two dimensional liquid jets and the varicose and sinuous modes are predicted [2]. The growth rate of initial disturbances for various azimuthal modes has been presented in a wide range of disturbances.

[1]. Yang, H. Q. 1992 Asymmetric instability of a liquid jet. Phys. Fluids **4**, 681-689.
[2]. Hagerty, W. W., Shea, J. F. 1955 A study of the stability of plane fluid sheets. J. Appl. Mech. **22**, 509-514.

[1]PhD Candidate
[2]Associate Professor

1:29PM R25.00004 The vortex breakdown of a variable property jet with swirling flow. , AHMAD ADZLAN FADZLI BIN KHAIRI, HIROSHI GOTODA, Ritsumeikan University, TOSHIHISA UEDA, Keio University — The vortex breakdown of a coaxial variable property jet with swirling flow has been experimentally investigated in this work, focusing on how the swirl of the inner and outer jets affect the formation of a stagnation point in the swirling jet. In the case of the CO_2 jet, the stagnation point flow is more easily formed compared to the air jet, and the stagnation point location was lower than that of the air jet. Stagnation point flow is also formed easier with the introduction of the swirl of the outer jet, and its location is also lower compared to the nonswirling case. The lowering of the stagnation point location of the swirling inner jet with density and viscosity differences due to the swirl of the inner and outer jets will be physically explained in this presentation by considering the theoretical equation obtained by analytically solving a simplified Navier-Stokes equation, (S. Matsubara, H. Gotoda, A. Adzlan, T. Ueda, Experiments in Fluids, 2011 (In press)) which has not been reported in previous research on fluid dynamics.

1:42PM R25.00005 Hydrodynamic Instabilities in Round Liquid Jets in Gaseous Crossflow , BARRY SCHARFMAN, ALEXANDRA TECHET, JOHN BUSH, MIT — Water jets in the presence of uniform perpendicular air crossflow were investigated theoretically and experimentally using high speed imaging for gaseous Weber numbers (We) below 30, small liquid jet Ohnesorge numbers, and large liquid and gaseous Reynolds numbers. Previously, a bag instability has been reported for We between 4 and 30. Jets first deform into curved sheets due to aerodynamic drag, followed by the formation of partial bubbles (bags) along the jet streamwise direction that expand and ultimately burst. Single bags were present at each streamwise position along the liquid jets in prior experiments featuring liquid jet nozzle diameters less than the capillary length of water. We have found that at larger nozzle diameters it is possible to observe multiple bags at the same streamwise jet position because single bags of such large sizes would be unstable. Theoretical predictions for individual bag expansion diameter over time agree with experimental measurements.

1:55PM R25.00006 Scalar and Velocity Field Measurements in Variable Density Jets in Crossflow[1] , DANIEL GETSINGER, LEVON GEVORKYAN, CORY HENDRICKSON, OWEN SMITH, ANN KARAGOZIAN, UCLA — This experimental study explores both unforced and acoustically forced behavior of variable density transverse jets via simultaneous acetone PLIF and PIV measurements. Jets composed of mixtures of helium and nitrogen are injected normally from a converging nozzle into an air crossflow. The jet-to-crossflow density ratio S is varied among test cases by changing the proportions of nitrogen and helium as well as the fraction of seeded acetone. A recent study[2] determined that transverse jets (of Reynolds number $Re_j = 1800$) likely transition to global instability in response to sufficient lowering of the jet-to-crossflow density ratio S (below 0.45-0.40) and/or momentum flux ratio J (below 10). This transition is characterized by weak shear layer instabilities that are easily overcome by external forcing for the convectively unstable (high S and J) case, and strong pure-tone oscillations resistant to external forcing for the globally unstable (low S and J) case. The effect of this instability transition on jet dynamics and mixing is examined here, as are alterations in the velocity field that may be associated with the behavior of the instabilities.

[1]Supported by NSF grant CBET-0755104 and the NASA GSRP.
[2]Getsinger, et al., AIAA Paper 2011-0040, 2011

2:08PM R25.00007 Analysis of nonlinear interactions among instability mechanisms in a jet in crossflow , MILOŠ ILAK, PHILIPP SCHLATTER, SHERVIN BAGHERI, DAN HENNINGSON, Linne Flow Centre and Swedish e-Science Research Centre, KTH Mechanics, SE-100 44 Stockholm, Sweden — We undertake an analysis of datasets from direct numerical simulation of a jet in crossflow, using the method of Dynamic Mode Decomposition (DMD). The procedure reveals coherent structures in the flow known as Koopman modes, which oscillate at frequencies that are also computed by the method. Both the crossflow and the jet inflow profile are laminar. As the jet-to-crossflow velocity R is increased, we observe the breakdown of hairpin vortices characteristic for low values of R. Near-wall structures corresponding to low-frequency oscillations are revealed above $R = 1.5$ by the DMD analysis, and their interaction with modes on the jet trajectory results in complex flow patterns, which however retain spanwise symmetry. At $R = 2.5$ and higher, spanwise symmetry is broken, and the flow exhibits the complex dynamics typically observed in literature. Furthermore, we study the effects of crossflow unsteadiness on jets at low R, showing that the hairpin vortices are able to persist under noise of moderate amplitude introduced upstream of the jet orifice.

2:21PM R25.00008 Linear forcing response of subsonic jets , XAVIER GARNAUD, LUTZ LESSHAFFT, PATRICK HUERRE, PETER SCHMID, LadHyX, CNRS / Ecole Polytechnique, France — The linear stability of spatially developing subsonic jets is investigated. A parametric base flow model is employed that matches experimental data for turbulent mean flows and that includes a solid nozzle. Temporal eigenmodes are computed using a newly developed "shift-relax" method. All eigenmodes are found to be stable in an isothermal setting. While this observation is in agreement with classical local stability results, a stable eigenmode spectrum seems inappropriate for the description of the convective instability dynamics of jets, which are known to be highly receptive to external perturbations. Instead, we propose to characterize jet instability in terms of the linear global flow response to sustained low-level forcing. External perturbations inside the nozzle duct are identified that give rise to the most amplified flow response at a prescribed frequency. Results will be discussed both for incompressible and compressible settings.

2:34PM R25.00009 Instability of low viscosity elliptic jets with varying aspect ratio , VARUN KULKARNI, Department of Mechanical Engineering, Purdue University — In this work an analytical description of capillary instability of liquid elliptic jets with varying aspect ratio is presented. Linear stability analysis in the long wave approximation with negligible gravitational effects is employed. Elliptic cylindrical coordinate system is used and perturbation velocity potential substituted in the Laplace equation to yield Mathieu and Modified Mathieu differential equations. The dispersion relation for elliptical orifices of any aspect ratio is derived and validated for axisymmetric disturbances with m =0, in the limit of aspect ratio, $\mu = 1$, i.e. the case of a circular jet. As Mathieu functions and Modified Mathieu function solutions converge to Bessel's functions in this limit the Rayleigh-Plateau instability criterion is met. Also, stability of solutions corresponding to asymmetric disturbances for the kink mode, $m = 1$ and flute modes corresponding to $m \geq 2$ is discussed. Experimental data from earlier works is used to compare observations made for elliptical orifices with $\mu \neq 1$. This novel approach aims at generalizing the results pertaining to cylindrical jets with circular cross section leading to better understanding of breakup in liquid jets of various geometries.

2:47PM R25.00010 Local absolute instability in the near field of hot and light round jets , WILFRIED COENEN, ALEJANDRO SEVILLA, Area de Mecanica de Fluidos, Universidad Carlos III de Madrid — We present a numerical investigation of the viscous spatiotemporal stability properties of low-density round jets emerging from circular nozzles or tubes. The two particular cases typically studied in experiments, namely a hot gas jet discharging into a quiescent cold ambient of the same species, and an isothermal jet consisting of a mixture between two gases with different molecular weight, discharging into a stagnant ambient of the heavier species, are treated separately. We use a realistic representation of the base velocity and density profiles based on boundary layer theory, taking into account the effect of variable transport properties. Our results show significant quantitative differences with respect to previous studies that use parametric presumed-shape base profiles, and reveal that hot jets are generically more unstable than light jets. In addition, the downstream evolution of the local stability properties of the jet is analyzed, revealing that, whereas a localized pocket of absolute instability can take place in the jet, for the combination of jet-to-ambient density ratio, Reynolds number, and initial momentum thickness used in experiments available in the literature, the absolutely unstable region in the jet is bounded by the jet outlet. The global transition observed in these experiments is demonstrated to take place when the absolutely unstable domain becomes sufficiently large.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R26 Nanofluids IV 329

12:50PM R26.00001 Electroosmosis in a nanofilm of chloride-aqueous solution with counter-charged surface patches[1] , HARVEY ZAMBRANO, MARIE PINTI, A.T. CONLISK, SHAURYA PRAKASH, The Ohio State University — We study Electroosmotic flow (EOF) by conducting Non-Equilibrium MD Simulations (NEMDS) of water and chloride on a silica substrate. The system response is studied as axial electric fields (AEF) are imposed and as the surface charge (SC) is modified by implementing counter-charged patches (CP). The density profiles reveal that the CP result in an ionic depletion in bulk solvent and in a higher hydrophilicity than on regular silica. We compute lower velocities for the cases with higher SC on the CP. Our velocities are in agreement to results from previous MD studies. Density and velocity profiles reveal a stationary chloride layer and a stagnant water region on the CP. This stationary layer grows as CP with higher charges are settled and as weaker AEF are imposed. We infer that ionic accumulation on the CP and hydrophilicity are responsible for the EOF velocity changes for systems with different CP and the same AEF imposed. We perform continuum calculations and the EOF velocities agree with the results obtained from NEMDS. We show that by modifying SC on a substrate, systematic changes can be induced in EOF at the nanoscale.

[1]Partially supported by the Army Research Office.

1:03PM R26.00002 Electrokinetics of Correlated Electrolytes and Ionic Liquids , BRIAN STOREY, Olin College, MARTIN BAZANT, MIT — Perhaps the most basic assumption of classical electrokinetic theory is the mean-field approximation, where the each ion feels only the electric field produced by the mean charge density (via Poisson's equation) rather than the fluctuating Coulomb forces with individual neighbors. Here, we present a simple continuum model for electrostatic correlations between finite-sized ions, which leads to a 4th order modified Poisson equation, convenient for the analysis of electrokinetic phenomena. When the mean-field approximation breaks down, e.g. due to large ion concentrations, large ion valences, and/or nanoscale confinement, the zeta potential loses its significance, and the model predicts that electro-osmotic flows are typically reduced - or even reversed - by correlation effects, compared to the prediction of the Helmholtz-Smoluchowski formula. This may help to explain the over-prediction of induced-charge electro-osmotic flows by classical models. An interesting limit of the model describes electro-osmosis in solvent-free ionic liquids and molten salts, which may be important in energy storage and electroactuation applications.

1:16PM R26.00003 Characterization of Electrical Properties of Nanowires by Electro-orientation , CEVAT AKIN, JERRY SHAN, Rutgers University — We investigate the electro-orientation of large-aspect-ratio particles in liquid suspension as a possible technique to determine the particles' electrical conductivity and/or permittivity, which are often poorly known and difficult to measure directly. With the application of a spatially uniform AC electric field, ellipsoidal particles in suspension will rotate into alignment with their longest axis along the field. In the low frequency limit, the alignment rate itself does not distinguish between equally sized particles of different properties. However, it is possible to characterize the particle's electrical properties by measuring the crossover frequency at which its alignment rate transitions from conductivity-dominated to permittivity-dominated behavior. Moreover, the crossover frequency is insensitive to the particle aspect ratio for large aspect ratios, making the electro-orientation technique suitable even for nanowires, whose precise length is difficult to resolve optically. We present experimental results obtained by optical microscopy on the alignment rate of nano- and micro- wires under applied fields of different frequency. Experiments are conducted with particles of different type and size to determine how the electro-orientation crossover frequency varies with particle conductivity and aspect ratio. We compare our experimental results with theoretically obtained values, and assess electro-orientation as a nanowire-characterization technique.

1:29PM R26.00004 Examining Permittivity Effects in Electric Double Layers using Molecular Dynamics and Atomistic-to-Continuum Modeling , JEREMY TEMPLETON, KRANTHI MANDADAPU, JONATHAN LEE, REESE JONES, JONATHAN ZIMMERAN, Sandia National Laboratories — Charged surfaces exposed to an ionic solution attract an electric double layer: stratifications of solvent and solute molecules significantly deviating from their bulk concentrations. The double layer screens the charges in solution from those at the surface, but due to the anisotropy in the near-wall fluid, characterizing the precise mechanics by which this happens has proven difficult. Molecular dynamics (MD) simulations are capable of explicitly resolving the complex layering effects in the vicinity of the charged interface, but much work remains to analyze the results and improve the understanding of these phenomena. In this work, we use atomistic-to-continuum methods to analyze the spatially-varying electrical permittivity in these layers at different potentials and ion concentrations, focusing on salt water solutions. The permittivity is a particularly relevant quantity because it determines the capacitance of the different ionic layers. We will present a mathematical formalism for extracting this quantity from an MD realization and provide examples of how this coarse-grained quantity varies with a simulation. It will then be demonstrated how the resolved permittivity informs the realized voltage drop across the various parts of the double layer.

1:42PM R26.00005 Atomistic simulations of nanoscale electrokinetic transport[1] , JIN LIU, University of Pennsylvania, MORAN WANG, Tsinghua University, Beijing, China, SHIYI CHEN, Peking University, Beijing, China, MARK ROBBINS, Johns Hopkins University — An efficient and accurate algorithm for atomistic simulations of nanoscale electrokinetic transport will be described. The long-range interactions between charged molecules are treated using the Particle-Particle Particle-Mesh method and the Poisson equation for the electric potential is solved using an efficient multi-grid method in physical space. Using this method, we investigate two important applications in electrokinetic transport: electroosmotic flow in rough channels and electowetting on dielectric (EWOD). Simulations of electroosmotic and pressure driven flow in exactly the same geometries show that surface roughness has a much more pronounced effect on electroosmotic flow. Analysis of local quantities shows that this is because the driving force in electroosmotic flow is localized near the wall where the charge density is high. In atomistic simulations of EWOD, we find the contact angle follows the continuum theory at low voltages and always saturates at high voltages. Based on our results, a new mechanism for saturation is identified and possible techniques for controlling saturation are proposed.

[1]This work is supported by the National Science Foundation under Grant No. CMMI 0709187

1:55PM R26.00006 Molecular Dynamics Models of the Electric Double Layer for Large Zeta Potentials , JONATHAN W. LEE, Sandia National Laboratories, JEREMY A. TEMPLETON, ROBERT H. NILSON, STEWART K. GRIFFITHS, BRYAN M. WONG, ANDY KUNG — The Classical Poisson-Boltzmann (PB) theory for the electric double layer (EDL) breaks down at the nanoscale as zeta potential increases. The ability to accurately model the EDL for large potentials is important for engineering high energy storage devices. To better understand behavior at large potentials, various molecular dynamics (MD) models were developed. MD models range from an idealized Lennard-Jones ionic fluid between unstructured walls to a salt water solution between solid substrates. All models feature a bulk fluid region in order to obtain a reference state. Models are compared using charge density profiles, solvent and solute concentration profiles, and zeta potentials as metrics. Local polarization structure can be obtained from the salt water MD models. Despite its inability to capture these effects, the idealized model similarly deviates from PB theory at large potentials. Ion concentration and surface charge density are varied in a parametric study using the idealized model.

2:08PM R26.00007 Characterization of the Velocity and Power Consumption of Electroosmotic Flow in Nanochannels Using Numerical Simulations , JOSHUA D. SHAWALA, FRANCISCO J. DIEZ, Rutgers University — A numerical simulation of the electroosmotic flow in a nanochannel is performed. The study focuses on characterizing the velocity behavior when changes are applied to the zeta potential in the electrical double layer and the electric field. Similarly, the characteristics of power consumption are indentified by the ratio of convection to conduction current, which changes with flow velocity and ion concentration. In the 2D numerical simulations the electrostatic potential is obtained by the Poisson Boltzmann dilute solution theory which is solved in its nonlinear form and is validated from published work. Aqueous solutions of 1:1 electrolytes at bulk ion concentrations from 1-100 mM are considered. The viscous-driven nature of the flow outside the electric double layer causes an adverse pressure gradient along the center of the channel which is examined in detail. Pressure effects at the inlet and outlet are also considered. The need for higher inlet pressure increases with velocity, which is proportional to the applied electric field. Increasing the bulk ion concentration causes minimal change in average velocity under conditions of zero slip at the channel walls. In the case when a slip length is imposed at the walls, the flow velocity increases significantly with ion concentration. The increase in slip length also increases the ratio of convection to conduction current.

2:21PM R26.00008 Lattice Boltzmann simulation of electrostatic double layer interaction force for nanoparticles[1] , GRACE X. SHI, YAN JIN, VOLHA LAZOUSKAYA, CHAO WANG, LIAN-PING WANG, University of Delaware — Modeling the transport and retention of nanoparticles (NPs) through soil porous media requires an accurate description of the electrostatic interaction force between a nanoparticle and soil grain. In this study, we apply the lattice Boltzmann method to directly solve the nonlinear Poisson Boltzmann (PB) equation for several geometric configurations including plate-plate, NP-plate, and NP-NP interactions, for any surface potentials and interaction distances and for different boundary conditions. Interaction energy and force are then derived from the simulations. For the case of plate-plate interaction, the simulation results are compared to the exact solution of the nonlinear PB equation. It is shown that the linear PB solution is valid when the nondimensional surface potential is less than one, and that the linear PB solution over-predicts the interaction force for intermediate gap distances but under-predicts the force for small gap distances. For NP-plate and NP-NP interactions, an axisymmetric lattice Boltzmann formulation is developed to solve the governing equations. The results will be compared to the classic approximate expressions of interaction force to evaluate their validity and to study the effect of nanoparticle size.

[1]Work supported by NSF and USDA.

2:34PM R26.00009 The effects of external electric field on the structure of water inside and outside single-walled carbon nanotubes[1] , ZHEN XU, GUO-HUI HU, ZHE-WEI ZHOU — In the present work, all-atom molecular dynamics (MD) simulations are utilized to examine the structure of water inside and outside the armchair SWCNT in the presence of external electric field parallel or perpendicular to the tube axis. Extensive MD simulations have been performed in wide ranges of E (0-3V/nm) at room conditions (300 K and 1 bar). The dependence of liquid density profile, orientation of dipole moment and hydrogen bonds profiles are discussed on the electric fields. With the parallel electric field, the structure of water outside the SWCNT changes slightly while inside the SWCNT the water structure is found to be more ordered. With the perpendicular electric field, the structure of water both inside and outside the SWCNT has changed dramatically. When the strength of field is above 1V/nm, the chains or the layers structures inside the SWCNT are broken and even the isolated water molecule is found. Outside the SWCNT, liquid density profiles, orientation of dipole moment and hydrogen bonds profiles are found to be the non-axisymmetric. This work may be helpful in understanding the physics of the confined water and in the design of future nanofluidic devices.

[1]The work was supported by the National Science Foundation of China (Grant No. 10872122).

2:47PM R26.00010 Modeling of Ion Transport in Nanochannels , JUSTYNA CZERWINSKA, Artorg Center for Biomedical Engineering Rsearch, Bern University, Switzerland — Transport of ions in fluidic environment is a basis for many biological processes. Microchannel ions flow is characterized by formation of electric double layer (EDL) near the solid wall. In nanochannel, however, the formation of EDL can be prevented by the constrained geometry leading to the creation of a single ion layer and resulting in the selective movement of ions. The confinement effects can be controlled by the wall charges providing a controlability to the nanoscale diffusion (pumping effect). This study will present model and molecular dynamics simulations of three-dimensional nanochannel flow of charged fluid. The various concentration of ions in solution was studied as well as the influence of the external force.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R27 Biofluids: General II: Microbubbles and Droplets Ballroom I

12:50PM R27.00001 The Acoustic Atomization of Droplets within a Bubble[1] , DAVID LI, ROBINSON SEDA-PADILLA, J. BRIAN FOWLKES, JOSEPH BULL, University of Michigan — The process of vaporizing liquid microdroplets using ultrasound is known as acoustic droplet vaporization (ADV). Gas embolotherapy is a proposed cancer therapy that uses the ADV process to selectively generate microbubbles, which can then lodge in the microvasculature to occlude blood flow and starve the tumor. We have observed that continued ultrasound exposure to microbubbles adhering to a wall induces in a droplet atomization process. The atomization process originates at the gas-liquid interface and produces a spray of liquid droplet within the microbubble along the axis of the acoustic beam. Single pulses with 30 cycles from 3.5 and 7.5 MHz single element focused transducers operating at peak negative pressures ranging from 4 to 8 MPa were used to generate atomization. The atomization process was observed in microbubbles ranged from 30 μm to 1 mm in diameter. The extent of the atomization had a direct relationship with acoustic pressure.

[1]This work is supported by NIH grant R01EB006476.

1:03PM R27.00002 Bubble Transport and Splitting in a Symmetric Bifurcation[1] , ADNAN QAMAR, JOSEPH BULL, University of Michigan — Transport and splitting of gas bubbles through a geometrically symmetric bifurcation is investigated numerically as a model of cardiovascular gas bubble transport in air embolism and Gas Embolotherapy. An interface capturing Volume of Fluid Method on an Eulerian fixed grid is used to compute the bubble splitting at the symmetric bifurcation. Bubble transport and splitting is investigated for a range of roll angles, capillary numbers, Reynolds numbers and Bond numbers. Results indicate that splitting is observed to be more homogenous at higher capillary numbers and lower roll angles. It is observed that at nonzero roll angles and small bubble lengths, there is a critical value of the capillary number below which the bubbles do not split and are transported entirely into the upper branch. The value of the critical capillary number increases with roll angle and the bubble length. Shear stress distribution at the bifurcation carina increases several folds as the bubble tip reaches the carina. These findings suggest that, in large vessels, gas emboli tend to be transported upward unless flow is unusually strong. In smaller vessels more even splitting of bubbles is predicted. The endothelial cells at a vessel bifurcation would be potentially exposed to higher stress levels, which might induce bioeffects.

[1]This work was funded by NIH grant R01EB006476.

1:16PM R27.00003 Frequency dependent subharmonic threshold for contrast microbubbles[1] , AMIT KATIYAR, KAUSIK SARKAR, University of Delaware — We numerically investigate the predictions from several contrast microbubble models to determine the excitation threshold for subharmonic generation. All models are transformed into a common interfacial rheological form, where encapsulation is represented by two radius dependent surface properties—effective surface tension and surface dilatational viscosity. In contrast to the classical perturbative result, the minimum threshold for subharmonic generation is not always obtained near twice the resonance frequency; instead it can occur over a range of frequency from resonance to twice the resonance frequency. The quantitative variation of the threshold with frequency depends on the model, bubble radius and encapsulation properties. Some models incorporate an upper limit on effective surface tension (resulting from strain softening or rupture of the encapsulation during expansion). Without this upper limit, the threshold is extremely large especially near the resonance frequency and there is a global minimum near twice the resonance frequency. On the other hand, having zero surface tension in the buckled state (the lower limit) increases the threshold especially near twice the resonance frequency which in presence of the upper limit results in a possible shift of the minimum threshold towards resonance.

[1]Partially supported by NSF and NIH.

1:29PM R27.00004 The fluid mechanics of nutrient transport within biofilms , MICHAEL BRENNER, AGNESE SEMINARA, NAVEEN SINHA, JAMES WILKING, Harvard University, TOMMY ANGELINI, University of Florida, ROBERTO KOLTER, DAVID WEITZ, Harvard University — Bacterial biofilms are interface-associated colonies of bacteria embedded in an extracellular matrix that is composed primarily of polymers and proteins. During the growth of a biofilm, nutrient is taken up by the surface of the biofilm, and contained by cells in the bulk. A critical problem is that above a critical size there is necessarily a growth bottleneck, in which the biofilm cannot take up enough nutrients to feed all of the cells within it. We discuss, through theory and experiments, several strategies that are employed by biofilms of Bacillus subtilus to avoid this growth bottleneck. These include clever use and control of osmotic pressure (through the expression of polymeric extracellular matrix); the excretion of surfactants and the use of associated marangoni stresses; and the distribution of flagella (used as mixers) within the bulk of the biofilm. Some speculations about other potential mechanisms (for which there is no current experimental support) will also be presented.

1:42PM R27.00005 Modeling and 3-D Simulation of Biofilm Dynamics in Aqueous Environment , QI WANG, Univ. of South Carolina — We present a complex fluid model for biofilms growing in an aqueous environment. The modeling approach represents a new paradigm to develop models for biofilm-environment interaction that can be used to systematically incorporate refined chemical and physiological mechanisms. Special solutions of the model are presented and analyzed. 3-D numerical simulations in aqueous environment with emphasis on biofilm-ambient fluid interaction will be discussed in detail.

1:55PM R27.00006 Variation of flow-induced stresses within scaffolds used in bone tissue engineering[1] , DIMITRIOS PAPAVASSILIOU, NGOC PHAM, ROMAN VORONOV[2], VASSILIOS SIKAVITSAS, The University of Oklahoma — Bone tissue engineering is often based on seeding adult stem cells on porous scaffolds and subsequently placing these scaffolds in flow perfusion bioreactors to stimulate cell differentiation and cell growth. In the present study, the distribution of stresses in structured porous scaffolds under flow is investigated by calculating the probability density function of flow-induced stresses in different scaffold geometries with simulations. The physical reason for the development of particular stress distributions is further explored, and it is found that the direction of flow relative to the internal architecture of the porous scaffold is important for stress distributions. When the flow direction is random relative to the configuration of the geometric elements making up the scaffold, it is found that a common distribution, such as the one suggested by Voronov et al. (Appl. Phys. Let., 2010, 97:024101), can be used to describe the stress distribution.

[1]NSF CBET-070081
[2]Currently at U Penn

2:08PM R27.00007 From thermodynamics to fluid mechanics: Enhancing the 2-D protein crystallization process

, JAMES YOUNG, DAVID POSADA, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — The leading method to determine protein structure is to perform X-ray diffraction on protein crystals. However, crystallizing protein is a challenging task which is usually met with limited success. 2-D protein crystals at the air/water interface are commonly obtained under quiescent conditions. Recently, the formation of such crystals was shown to be enhanced by the presence of flow. Here, we examine both the kinetics of the process, including the detrimental effects of protein aggregation, as well as the fluid dynamics associated with successful crystallization events. The deep-channel surface viscometer geometry is utilized which consists of the annular flow between two stationary cylinders and a rotating floor. For a particular protein surface concentration, a Reynolds number threshold has been identified above which crystals grow and below which they do not. This flow system also allows for the determination of the surface shear viscosity, which provides an indication of the mesoscale interactions associated with protein crystals.

2:21PM R27.00008 Constrained droplets for high resolution microscopy of protein fibrillization

, DAVID POSADA, PETER TESSIER, AMIR HIRSA, Rensselaer Polytechnic Institute — The use of constrained droplets (droplets with pinned contact lines on solid surfaces) is proposed here as a method for sample support in optical microscopy studies. Capillarity acts to contain the liquid sample, allowing access for observations in the bulk and at the gas/liquid interface. At the capillary length scale, surface tension forms stable interfaces, virtually immune to gravity and with curvatures that can be adjusted. This is particularly useful when studying the gas/liquid interface and its vicinity under high resolution optical microscopy. Such observations are normally performed using oil immersion objectives which must be positioned within distances only tens of microns from the region of interest. Constrained droplets can also be used at small scales, requiring minute volumes of analyte. The use of the constrained droplet method is demonstrated by studying the aggregation of insulin into amyloid fibrils in the solution and at the gas/liquid interface, where proteins are prone to denaturation and subsequent fibrillization. Such an aggregation process is associated with many neurodegenerative diseases, including Alzhemier's.

2:34PM R27.00009 Material characterization of poly-lactic acid shelled ultrasound contrast agent and their dynamics[1]

, SHIRSHENDU PAUL, DANIEL RUSSAKOW, TYLER RODGERS, KAUSIK SARKAR, University of Delaware, MICHAEL COCHRAN, MARGARET WHEATLEY, Drexel University — Micron-size gas bubbles encapsulated with lipids and proteins are used as contrast enhancing agents for ultrasound imaging. Biodegradable polymer poly-lactic acid (PLA) has recently been suggested as a possible means of encapsulation. Here, we report *in vitro* measurement of attenuation and scattering of ultrasound through an emulsion of PLA agent as well as theoretical modeling of the encapsulated bubble dynamics. The attenuation measured with three different transducers of central frequencies 2.25, 3.5 and 5 MHz, shows a peak around 2-3 MHz. These bubbles also show themselves to possess excellent scattering characteristics including strong non-linear response that can be used for harmonic and sub-harmonic contrast imaging. Our recently developed interfacial rheological models are applied to describe the dynamics of these bubbles; rheological model properties are estimated using measured attenuation data. The model is then applied to predict nonlinear scattered response, and the prediction is compared against experimental observation.

[1]Partially supported by NSF and NIH

2:47PM R27.00010 Blood Flow through an Open-Celled Foam

, JASON ORTEGA, LLNL, DUNCAN MAITLAND, Texas A&M University, LLNL — The Hazen-Dupuit-Darcy (HDD) equation is commonly used in engineering applications to model the pressure gradient of flow through a porous media. One major advantage of this equation is that it simplifies the complex geometric details of the porous media into two coefficients: the permeability, K, and form factor, C. However through this simplification, the flow details within the porous media are no longer accessible, making it difficult to study the phenomena that contribute to changes in K and C due to clotting of blood flow. To obtain a more detailed understanding of blood flow through a porous media, a direct assessment of the complex interstitial geometry and flow is required. In this study, we solve the Navier-Stokes equations for Newtonian and non-Newtonian blood flow through an open-celled foam geometry obtained from a micro-CT scan. The nominal strut size of the foam sample is of O(10e-5) m and the corresponding Reynolds number based upon this length ranges up to $O(10)$. Fitting the pressure gradient vs. Darcy velocity data with the HDD equation demonstrates that both viscous and inertial forces play an important role in the flow through the foam at these Reynolds numbers. Recirculation zones are observed to form in the wake of the pore struts, producing regions of flow characterized by both low shear rates and long fluid residence times, factors of which have been shown in previous studies to promote blood clotting.

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R28 Flapping Flight Ballroom II

12:50PM R28.00001 Time-resolved measurements of the velocity field over the wing of bats during flight

, COSIMA SCHUNK, SHARON SWARTZ, KENNETH BREUER, Brown University — Particle Image Velocimetry (PIV) has become a well-established tool to study flows associated with flying animals. The wake shed by flying bats as it is seen in the Trefftz plane is by now well described for several bat species. However, to complete the understanding of the three-dimensional wake structures, additional views are necessary. To meet this need, bats were trained to fly at a stationary position in the wind tunnel at wind speeds between 2 m/s and 6 m/s. Aligning the laser light sheet parallel to the free stream, we measured, using time-resolved PIV (200 Hz), the air flow in the region of the left wing of the animals. Three high-speed cameras (400 Hz) were used to capture the position and movement of the bat and to reconstruct the wing kinematics. Characteristic flow structures were observed consistently at different spanwise positions, including the starting and stopping vortices at the beginning and end of the downstroke, as well as other vortex and wake features.

1:03PM R28.00002 Inertial and Fluid Forces during Bat Flight Maneuvers

, ATTILA BERGOU, JENNIFER FRANCK, GABRIEL TAUBIN, SHARON SWARTZ, KENNETH BREUER, Brown University — Flying animals generate forces to move through the air with the coordinated movement of their wings. Bats have evolved a particularly impressive capacity in flight control. With more than 24 wing joints bats are able to manipulate wing area, angle of attack, and camber to control their flight through altering aerodynamic forces on their wings. Here we use a model-based tracking framework to reconstruct the highly articulated wing and body kinematics of maneuvering bats from high-speed video. Using this data, we extract a simplified wing geometry and kinematics during various flight maneuvers. The time-dependent fluid flow and resultant forces on the wing are computed with CFD using a direct numerical simulation, and then used in a low-order model of the bat dynamics. This reconstruction identifies the relative importance of both inertial and aerodynamic forces during flight maneuvers.

1:16PM R28.00003 Large-Eddy Simulations of Flapping-Induced Lift Enhancement

, JENNIFER FRANCK, SHARON SWARTZ, KENNETH BREUER, Brown University — This work isolates the heaving motion of flapping flight in order to numerically investigate the fluid-structure interaction at Reynolds numbers relevant to birds and bats. Although there has been much focus on insect flight, larger vertebrates fly at a higher Reynolds number, which leads to different dynamics in terms of flow separation, reattachment, and high-lift mechanisms. In this work, an incompressible large-eddy simulation is used to simulate the periodic heaving of a flat plate at various angles of attack. It is found that the heaving motion can increase the average lift when compared with the steady flow, more so than is expected from the relative angle of attack. The additional lift is attributed to the vortex dynamics at the leading edge. The lift enhancement and flow features are compared with experimental results.

1:29PM R28.00004 Lift force enhancement and fluid-structure interactions on a self-excited flapping wing model[1] , OSCAR CURET, SHARON SWARTZ, KENNETH BREUER, Brown University — We present data from a mechanical model that we have used to explore a physical mechanism that may have aided transition from gliding to flapping flight over fifty million years ago. The model is composed of a cantilevered flat plate with a hinged trailing flap and is tested in a low-speed wind tunnel. For slow wind speeds the model is stationary, but above a critical wind speed the wing starts to oscillate due to an aeroelastic instability. A positive angle of attack on the wing results in a positive lift force. However, this lift force is significantly enhanced once the wing starts to oscillate. We used particle image velocimetry (PIV) to understand the unsteady aerodynamics of the self-excited flapping wing, and to identify and characterize the mechanisms that generate the enhanced lift force. We also discuss the implications of our results on the evolution of powered biological flight.

[1]This work was supported by AFOSR-MURI on bioinspired flight.

1:42PM R28.00005 Fluid-structure interactions on compliant membrane wings , RYE WALDMAN, SHARON SWARTZ, KENNETH BREUER, Brown University — Membrane wings are characteristic of flying animals such as bats, as well as low Reynolds number Micro Air Vehicles. These wings exhibit interesting features such as self-cambering, soft stall, and good performance at large angles of incidence. The interaction between the membrane and the vortical structures over the wing play an important role in the wing's performance. Vortices shed from the leading edge and tip interact with the membrane to select vibration modes, which depend on the details of the wing loading. However, the vibration modes are sensitive to the boundary conditions of the membrane. Here, we present results using force and membrane displacement measurements as well as Particle Image Velocimetry (PIV) from wind tunnel experiments on rectangular membrane wings with identical planform, but with different kinds of perimeter support. The different boundary conditions on the membrane affect the wing shape, support different vibration modes at different frequencies, and affect aerodynamic performance.

1:55PM R28.00006 A bi-directional leading-edge vortex in slow-flying bats , SHIZHAO WANG, LNM, Institute of Mechanics, Chinese Academy of Sciences, XIN ZHANG, GUOWEI HE, LNM, Institute of Mechanics, Chinese Academy of Science — A leading-edge vortex (LEV) is crucial to bat afloat, since a LEV could generate high lift which could not be predicted by the conventional aerodynamics theories. The LEV usually exhibits an intensive spiral vortex of a unidirectional axial flow on the top surface of wing. In this study, we numerically simulate a slowing-flying bat using immersed boundary method. The morphology and kinematics of bat are taken from experimental measurements. It is observed from our simulation that the stretching and collapse motions of wing could induce a bi-directional axial flow. The bi-directional axial flows stabilize the LEV and enhance its intensity. The observation is further investigated by using a simple model: the flows around a spanwise oscillating plate. The spanwise oscillation could enhance the LEV and make its more stable. This result implies a link of bat kinematics with its unusual aerodynamic performances.

2:08PM R28.00007 Force estimation and turbulence in the wake of a freely flying European Starling , HADAR BEN-GIDA, BGU, ADAM KIRCHHEFER, GREGORY KOPP, UWO, ROI GURKA, BGU — Flapping wings are one of the most complex yet widespread propulsion method found in nature. Although aeronautical technology has advanced rapidly over the past 100 years, natural flyers, which have evolved over millions of years, still feature higher efficiency and represent one of nature's finest locomotion methods. One of the key questions is the role of the unsteady motion in the flow due to the wing flapping and its contribution to the forces acting on a bird during downstroke and upstroke. The wake of a freely flying European Starling is investigated as a case study of unsteady wing aerodynamics. Measurements of the near wake have been taken using long duration high-speed PIV in the wake behind a freely flying bird in a specially designed avian wind tunnel. The wake has been characterized by means of velocity and vorticity fields. The measured flow field is decomposed based on the wing position phases. Drag and lift have been estimated using the mean velocity deficit and the circulation at the wake region. In addition, kinematic analysis of the wing motion and the body has been performed using additional high-speed cameras that recorded the bird movement simultaneously with the PIV. Correlations between the wing kinematics and the flow field characteristics are presented as well as the time evolution of the velocity, vorticity and additional turbulence parameters.

2:21PM R28.00008 Characterization and Scaling of Vortex Shedding from a Plunging Plate[1] , AZAR ESLAM PANAH, JAMES BUCHHOLZ, University of Iowa — Leading-edge and trailing-edge vortices (LEV and TEV) are investigated for a plunging flat plate airfoil at a chord Reynolds number of 10,000 while varying plunge amplitude and Strouhal number. Digital Particle Image Velocimetry is used to examine the strength and dynamics of shed vortices. Vortex strength, timing, pinch-off and trajectory are examined. By tracking the development of both the LEV and TEV in phase-locked measurements throughout the cycle and extracting the respective vortex circulation, the dimensionless circulation of both the LEV and TEV at each phase in the cycle could be determined. Guided by theoretical considerations for vorticity generation and aerodynamic theory, we will discuss the role of kinematic parameters on vortex shedding and the applicability of a scaling factor for the circulation of the shed vortex structures. Whereas a scaling parameter based on plate kinematics effectively collapses the circulation values of the shed leading-edge vortices with variation in Strouhal number, plunge amplitude, and angle of attack, it is found that the strength of the trailing-edge structures vary little with variation in plunge amplitude and angle of attack.

[1]This work is supported by AFOSR under award number FA9550-11-1-0019 monitored by Dr. Doug Smith.

2:34PM R28.00009 Optimization of Flapping Based Locomotion , SHAWN WALKER, Louisiana State University, MICHAEL SHELLEY, New York University, Courant Institute — Locomotion at the macro-scale is important in biology and industrial applications, such as for understanding the fundamentals of flight to enable design of artificial locomotors. We present results on optimal actuation profiles for locomotion of a rigid, flapping body at intermediate Reynolds number. The actuation consists of a vertical velocity control attached to a pivot point of an ellongated rigid body, which is allowed to rotate and is affected by a torsional spring; the spring acts as an elastic recoil. No a priori assumption is made on the form of the vertical actuation, except for smoothness. Thus, we pose an infinite dimensional time-varying, PDE-constrained optimization problem (with additional constraints on the vertical control) and solve it by variational methods. We explore the effects of parameter variations on the optimal locomotion profile, such as the torsional spring constant, relative mass density of body to fluid, and discuss the effects on locomotion strategies.

2:47PM R28.00010 Spontaneous motion of flapping wings driven by hydrodynamic instability , OLIVIER MARQUET, ONERA, JUAN GUZMAN INIGO — Recent experimental [1] and numerical [2] studies have examined the dynamics of rigid symmetric wings flapping vertically in a quiescent fluid and free to move in the horizontal direction. It has been observed that above a critical flapping frequency the flow loses its symmetry while the wing starts to move horizontally and eventually reaches a quasi constant horizontal speed. The present work reconsiders this problem from an hydrodynamics instability point of view. The basic flow is periodic and symmetric, the periodicity being imposed by the vertical forcing frequency while the symmetry of the velocity field ensures no motion in the horizontal direction. The linear stability is examined using the Floquet theory, with the assumption of asymmetric perturbations to explain the onset of horizontal forces. Numerical results of the stability problem will be shown. The cases of wings fixed or free to move will be analyzed and compared.

[1] Vandenberghe N., Zhang J. & Childress S., "A symmetry-breaking leads to forward flapping", Journal of Fluid Mechanics, 506, 147 (2004)
[2] Alben S. & Shelley M. "Coherent locomotion as an attracting state for a free flapping body", Proc. Natl. Acad. Sci., U.S.A., 102, 11163 (2005)

Tuesday, November 22, 2011 12:50PM - 3:00PM –
Session R29 Suspension Swimming Ballroom III

12:50PM R29.00001 Out-of-Equilibrium effects in suspensions of light-activated artificial microwimmers , JEREMIE PALACCI, STEFANO SACANNA, DAVID PINE, PAUL CHAIKIN, CSMR, NYU, CSMR, NYU TEAM — We present a new type of colloidal artificial swimmers propelled by the decomposition of hydrogen peroxide and activated by light. The effect is reversible and allows external control of the swimming/non swimming behavior of the particles. Moreover the bulk synthesis makes possible the study of very concentrated assemblies of monodisperse and identical microswimmers. These active agents are used to explore fancy effects of out-of-equilibrium systems, e.g. clustering, "tugboat" effect...

1:03PM R29.00002 Laboratory studies of ocean mixing by microorganisms , MONICA MARTINEZ-ORTIZ, Mechanical Engineering, California Institute of Technology, JOHN O. DABIRI, Graduate Aeronautical Laboratories and Bioengineering, California Institute of Technology — Ocean mixing plays a major role in nutrient and energy transport and is an important input to climate models. Recent studies suggest that the contribution of fluid transport by swimming microorganisms to ocean mixing may be of the same order of magnitude as winds and tides. An experimental setup has been designed in order to study the mixing efficiency of vertical migration of plankton. To this end, a stratified water column is created to model the ocean's density gradient. The vertical migration of Artemia Salina (brine shrimp) within the water column is controlled via luminescent signals on the top and bottom of the column. By fluorescently labelling portions of the water column, the stirring of the density gradient by the animals is visualized and quantified. Preliminary results show that the vertical movement of these organisms produces enhanced mixing relative to control cases in which only buoyancy forces and diffusion are present.

1:16PM R29.00003 A kinetic model for concentrated active suspensions , BARATH EZHILAN, DAVID SAINTILLAN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, Urbana, Illinois 61801, USA, MICHAEL SHELLEY, Courant Institute of Mathematical Sciences, New York University, New York, New York 10012, USA — We study the dynamics in concentrated suspensions of active self-propelled particles using a kinetic model and three-dimensional continuum simulations.In a previous model for semi-dilute active suspensions (Saintillan and Shelley 2008), we have revealed the existence of hydrodynamic instabilities and complex correlated motions with pattern formation in suspensions of pushers, while suspensions of pullers undergo no such dynamics. Here, we modify this previous kinetic model by including an additional nematic alignment torque proportional to the local concentration in the equation for the rotational velocity of the particles, causing them to align locally with their neighbors (Doi and Edwards 1986). Linear stability analyses in the isotropic and aligned states show that in the presence of such a torque both pusher and puller suspensions are unstable to random fluctuations. Large-scale three-dimensional simulations of the kinetic equations are also performed and confirm the existence of these instabilities, which lead to the formation of nematic-like structures at high concentrations. Detailed measures are defined to quantify the degree and direction of alignment, and the effects of steric interactions on pattern formation are discussed in detail.

1:29PM R29.00004 Instabilities and global order in concentrated suspensions of spherical microswimmers , ARTHUR EVANS, UCLA, TAKUJI ISHIKAWA, TAKAMI YAMAGUCHI, Tohoku University, ERIC LAUGA, UCSD — We use numerical simulations to probe the dynamics of concentrated suspensions of spherical microswimmers interacting hydrodynamically. Previous work in the dilute limit predicted instabilities of aligned suspensions for both pusher and puller swimmers, which we confirm computationally. Unlike previous work, we show that isotropic suspensions of spherical swimmers are also always unstable. Both types of initial conditions develop long-time polar order, of a nature which depends on the hydrodynamic signature of the swimmer but very weakly on the volume fraction up to very high volume fractions.

1:42PM R29.00005 Transverse swimming in a dilute suspension of active swimmer under an oscillating shear flow , FRANCISCA C. GUZMAN LASTRA, RODRIGO SOTO, Universidad de Chile — Simulations of a dilute suspension of pusher swimmers under an oscillating shear flow show that a large fraction of them swim preferentially in the vortex and flow direction, perpendicular to the gradient direction. These two directions alternate in a collective way among all the swimmers. Experiments of a suspension of E.Coli in a Hele-Shaw cell under an oscillating flow manifest the same behavior.

1:55PM R29.00006 Effect of viscoelasticity on the collective behavior of swimming microorganisms , PATRICK UNDERHILL, YASER BOZORGI, Rensselaer Polytechnic Institute — Hydrodynamic interactions of swimming microorganisms can lead to coordinated behaviors of large groups. Previous theoretical work has shown that in mean-field theories the interactions give rise to a linear instability of the uniform isotropic state if the organisms push themselves forward. However, that work is done with the organisms suspended in a Newtonian fluid, while many organisms exist in a non-Newtonian medium. Using a mean-field theory and the Oldroyd-B constitutive equation, we show how viscoelasticity of the suspending fluid alters the hydrodynamic interactions and therefore the ability of the group to coordinate. We quantify the ability to coordinate by the initial growth rate of a small disturbance from the uniform isotropic state. For small wavenumbers the response is qualitatively similar to a Newtonian fluid but the Deborah number determines the effective viscosity of the suspension. At higher wavenumber, the response of the fluid to small amplitude oscillatory shear flow leads to a maximal growth rate at a particular wavelength. This is in stark contrast to the Newtonian result.

2:08PM R29.00007 Shaken, but not stirred: how vortical flow drives small-scale aggregations of gyrotactic phytoplankton , MICHAEL BARRY, WILLIAM DURHAM, MIT, ERIC CLIMENT, IMFT, ROMAN STOCKER, MIT — Coastal ocean observations reveal that motile phytoplankton form aggregations at the Kolmogorov scale (mm-cm), whereas non-motile cells do not. We propose a new mechanism for the formation of this small-scale patchiness based on the interplay of turbulence and gyrotactic motility. Counterintuitively, turbulence does not stir a plankton suspension to homogeneity but drives aggregations instead. Through controlled laboratory experiments we show that the alga *Heterosigma akashiwo* rapidly forms aggregations in a cavity-driven vortical flow that approximates Kolmogorov eddies. Gyrotactic motility is found to be the key ingredient for aggregation, as non-motile cells remain randomly distributed. Observations are in remarkable agreement with a 3D model, and the validity of this mechanism for generating patchiness has been extended to realistic turbulent flows using Direct Numerical Simulations. Because small-scale patchiness influences rates of predation, sexual reproduction, infection, and nutrient competition, this result indicates that gyrotactic motility can profoundly affect phytoplankton ecology.

2:21PM R29.00008 Enhancement of biomixing by swimming cells in 2D films[1] , JERRY GOLLUB, HUSEYIN KURTULDU, Haverford College, JEFFREY GUASTO, Massachusetts Institute of Technology, KARL JOHNSON, Haverford College — Fluid mixing in active suspensions of microorganisms is important to ecological phenomena and shows surprising statistical behavior. We investigate the mixing produced by swimming unicellular algal cells (Chlamydomonas) in quasi-2D films by tracking the motions of cells and of microscopic passive tracer particles advected by the fluid. The reduced spatial dimension of the system leads to long-range flows and a surprisingly strong dependence of tracer transport on the swimmer concentration. The mean square displacements are well described by a stochastic Langevin model, with an effective diffusion coefficient D growing as the 3/2 power of the swimmer concentration, due to the interaction of tracer particles with multiple swimmers. We also discuss the anomalous probability distributions of tracer displacements, which become Gaussian at high concentration, but show strong power-law tails at low concentration.[2]

[1]Supported by NSF Grant DMR-0803153

[2]H. Kurtuldu et al., Proc. Nat. Acad. Sci. 108, 10391 (2011)

2:34PM R29.00009 Active stress driven convection in a suspension of chemotactic bacteria[1] , KASYAP T.V., DONALD KOCH, School of Chemical and Biomolecular Engineering, Cornell University — We examine the linear stability of a suspension of swimming bacteria producing dipolar hydrodynamic disturbances confined in a channel subjected to a linear chemo-attractant gradient across the channel. At the continuum level swimming bacteria exert an "active" stress on the fluid which is a function of the bacterial concentration and orientation fields. In the base-state without any fluid flow, the fluxes from the chemotactic and diffusive motion of the bacteria balance to yield exponential number density and active stress profiles across the channel. We show that such a base-state is unstable to perturbations in the number density parallel to the channel walls if the bacterial concentration exceeds a critical value determined by a Peclet number measuring the strength of chemotaxis relative to diffusion. Active stress gradients resulting from the perturbation in the number density drive convective fluid flow, which transports bacteria into the regions of highest perturbed bacteria concentration reinforcing the original perturbation.

[1]This work is supported by NSF grant CBET-0730579.

2:47PM R29.00010 Energy Transport in a Concentrated Suspension of Bacteria , TAKUJI ISHIKAWA, NAOTO YOSHIDA, HIRONORI UENO, MATTHIAS WIEDEMAN, YOHSUKE IMAI, TAKAMI YAMAGUCHI, Tohoku University — Coherent structures appear in a concentrated suspension of swimming bacteria. While transport phenomena in a suspension have been studied extensively, how energy is transported from the individual cell scale to the larger meso-scale remains unclear. In this study, we carry out the first successful measurement of the three-dimensional velocity field in a dense suspension of bacteria. The results show that most of the energy generated by individual bacteria dissipates on the cellular scale. Only a small amount of energy is transported to the meso-scale, but the gain in swimming velocity and mass transport due to meso-scale coherent structures is enormous. These results indicate that collective swimming of bacteria is efficient in terms of energy. This study sheds light on how energy can be transported toward smaller wave numbers in the Stokes flow regime.

Tuesday, November 22, 2011 12:50PM - 2:47PM –
Session R30 Microfluids: General VII Ballroom IV

12:50PM R30.00001 Tracking the Growth Rate of Nanopillar Formations Caused by Large Thermocapillary Forces , EUAN MCLEOD, SANDRA TROIAN, California Institute of Technology, 1200 E. California Blvd MC 128-95, Pasadena, CA — Viscous nanofilms are known to deform spontaneously into periodic arrays of nanopillars when exposed to a strong transverse thermal gradient. Comparison of the characteristic pillar spacing in experiment with predictions of linear instability analysis suggests that thermocapillary forces, and not acoustic phonon pressure or electrostatic image charge, are likely the dominant destabilizing mechanism [1]. Examination of the dynamical shapes and growth rates of emerging peaks provides an even more stringent test of the physical mechanism underlying the deformation process. Here we report measurements based on white light interferometry in which the reflected intensity from individual color channels is used to monitor the growth and shape of 3D formations in molten polymer nanofilms. Numerous experiments were conducted to isolate the influence of various operating parameters. These measurements exhibit an extended regime characterized by exponential growth which persists well beyond small amplitude deformations. The corresponding growth rates agree well with predictions of linear stability theory based on thermocapillary flow; however, the inferred film viscosities are systematically larger than bulk values.

[1] E. McLeod, Y. Liu, and S. M. Troian, Phys. Rev. Lett. 106, 175501(2011)

1:03PM R30.00002 Dynamics of flexible molecules in thinning fluid filaments , PAULO E. ARRATIA, GABRIEL JUAREZ, University of Pennsylvania — Newtonian liquids that contain small amounts ($\sim$ppm) of flexible polymers can exhibit viscoelastic behavior in extensional flows. In this talk, we report the results of experiments on the thinning and breakup of polymeric fluids in a simple microfluidic device. We aim to understand the stretching dynamics of flexible polymers by direct visualization of fluorescent DNA molecules, a model polymer. A Boger fluid, composed of 100 ppm polyacrylamide and 85% w/w glycerol, is seeded with stained lambda—DNA molecules ($<$ 10% v/v) imaged by high speed epifluorescence microscopy. We observe that the strong flow in the thinning fluid threads provide sufficient forces to stretch the DNA molecules away from their equilibrium coiled state. The distribution of stretch lengths, however, is very heterogeneous due to molecular individualism and initial conditions. Once the molecules are stretched to their full length and aligned with the flow, they translate along the fluid thread as rigid rods until the point of pinch off. After pinch off, both the fluid and molecules return to a relaxed state.

1:16PM R30.00003 Optimum Transport in Flat Heat Pipes , YU-WEI LIU, MARIN SIGURDSON, PAYAM BOZORGI, NOEL MACDONALD, CARL MEINHART, UCSB — In this study we investigate wetting behavior of the wick structure and the maximum theoretical heat transfer rate of a 40 cm titanium flat heat pipe. Large scale flat heat pipes are designed for high performance electronics cooling. Wick designs in flat heat pipes are typically limited by viscous drag and capillary pressure, and do not transport fluids as sufficient rates to meet practical cooling requirements. An analytical model is used to describe flow through wick structure with array of pillars. The capillary pressure and viscous drag are obtained by surface energy calculation and numerical simulations, respectively. To verify the model, we conducted wetting tests on the wick samples with different pillar dimensions. The model agrees qualitatively with the experiments, but under predicts the viscous drag. We extend the model to calculate the pressure drop for liquid and vapor flows, which predicts the upper limit for heat transport in flat heat pipes. Pillar parameters are optimized for maximum heat transfer rate, which approaches several hundred of Watts.

1:29PM R30.00004 Validation of CFD models for microscale nanoprecipitation reactor using μ-PIV and confocal μ-LIF

YANXIANG SHI, 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. — Over the past a few decades, computational fluid dynamics (CFD) models have become more and more important in the process of reactor design in chemical engineering. Compared to experimental methods, they can provide comprehensive information on the flow field as well as other fields, such as concentration. However, they also need to be validated against experimental data to ensure the accuracy. In this work, the micro-scale particle image velocimetry (μ-PIV) is employed in conjunction with the confocal-base micro-scale laser induced fluorescence (μ-LIF) to specifically validate CFD models for use in microscale nanoprecipitation reactor. The former is for the velocity field measurement and the latter gives us the mixture fraction information. Both RANS and LES are used to simulate the field flow. For RANS, a DQMOM-IEM micromixing model is used to predict the mixture fraction field while only a scalar transport equation is solved in the LES simulations. Comparisons between simulation results and experimental data show that RANS might not be the right tool for such reactors. LES, on the other hand, gives reasonably satisfactory predictions.

1:42PM R30.00005 Thermophoresis of a temperature responsive polymer[1]

JENNIFER KREFT PEARCE, KLINTON KILGORE, AUDREY HAMMACK, JACOB FORD, The University of Texas at Tyler — Thermophoresis, the migration of a species due to a temperature gradient, has been shown to be a possible mechanism for manipulating molecules in microfluidic devices. The mechanism governing thermophoresis is complex making its dependence on different physical factors hard to predict. We experimentally investigate thermophoresis of a polymer which exhibits inverse temperature dependence of its solubility in water. For sufficiently high average temperatures, two forms of the molecule are present. We measure the Soret coefficient of both and find that one has positive S_T and the other negative. We investigate the cause of this sign change using a Lattice Boltzmann based simulation. We find that the conformation of the polymer can influence its migration in a temperature gradient.

[1]Acknowledgment is made to the Donors of the American Chemical Society Petroleum Research Fund for support of this research.

1:55PM R30.00006 Microscale heat transfer enhancement using spinodal decomposition

PIETRO POESIO, DAFNE MOLIN, Università di Brescia, via Branze 38, 25123 Brescia, Italy, NICOLAS G. HADJICONSTANTINOU, Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, MA 02139, USA, GIAN PAOLO BERETTA, Università di Brescia, via Branze 38, 25123 Brescia, Italy — In many cases, miniaturization is limited by our ability to quickly remove heat; current state-of-the-art cooling approaches have significant limitations, particularly for high heat flux applications. Recent studies have shown that phase separation of a binary liquid-liquid mixture quenched to a temperature below the spinodal curve can be used to enhance heat transfer in small-scale devices. In particular, it has been shown that the self propulsion of single droplets formed during the intermediate stage of spinodal decomposition can produce considerable agitation and, as a result, enhanced heat transport. Spinodal phase separation dynamics can be described by the coupled Cahn-Hilliard/Navier-Stokes equations; unfortunately, simulation of these equations at the device scale is computationally costly due to the mulltiscale nature of spinodal decomposition, which requires resolution of the phase interface between the two fluids which is of atomistic size. In this talk we discuss possible approaches for reducing this computational cost by calculating the resulting transport from synthetic fluctuating fields that simulate the effect of spinodal decomposition but are generated stochastically without solving the Cahn-Hilliard equation at close-to-atomistic resolution.

2:08PM R30.00007 Improved theory on AC electrothermal flows

SOPHIE LOIRE, PAUL KAUFFMANN, IGOR MEZIC, UCSB — We compare simulations from new theory to experimental measurements on AC eletrothermal flows (ACET) for micromixing application on 96 microwell (10 μL) plate for high conductivity physiological solutions. This application leads to certain design constraints (electrode sizes, voltage range, conductivity). Beneath each microwell filled with saline solution (σ=0.02 mS/cm, to 16 mS/cm.), a sinusoidal voltage (0 to 40Vpp, 1MHz) is applied between 3 interdigitated gold electrodes 35 μm thick, separated by a 150μm gap. Due to this design, the ACET flows, measured by μPIV, doesn't follow the present theory. Similarly to natural convection, a bifurcation like behaviour is observed : the flows appear only above a critical voltage. The velocities scale as V^p with $p \geq 4$ with p increasing with conductivities. We analyse the validity conditions of the weak temperature gradient approximations. Accordingly we propose a thermal-electrical strong coupling model, which is traditionally neglected. We also study the competition between ACET and natural convection appearing in this configuration.

2:21PM R30.00008 Controlling the trajectories of bubble trains at a microfluidic junction

PRAVIEN PARTHIBAN, SAIF KHAN, National University of Singapore — The increasing number of applications facilitated by digital microfluidic flows has resulted in a sustained interest in not only understanding the diverse, interesting and often complex dynamics associated with such flows in microchannel networks but also in developing facile strategies to control them. We find that there are readily accessible flow speeds wherein resistance to flow in microchannels decreases with an increase in the number of confined bubbles present, and exploit this intriguing phenomenon to sort all bubble of a train exclusively into one of the arms of a nominally symmetric microfluidic loop. We also demonstrate how the arm into which the train filters into can be chosen by applying a *temporary* external stimulus by means of an additional flow of the continuous liquid into one the arms of the loop. Furthermore, we show how by tuning the magnitude and period of this temporary stimulus we can switch controllably, the traffic of bubbles between both arms of the loop even when the loop is *asymmetric*. The results of this work should aid in developing viable methods to regulate traffic of digital flows in microfluidic networks.

2:34PM R30.00009 Measurement of charge of a droplet induced by contact electrification

DONGWHI CHOI, HORIM LEE, Department of Mechanical Engineering, POSTECH, Korea, DO JIN IM, Department of Chemical Engineering, POSTECH, Korea, KWAN HYOUNG KANG, Department of Mechanical Engineering, POSTECH, Korea — The contact electrification is the charge transfer between two surfaces by contact and separation. We have developed the experimental method to measure the amount of charge of a droplet induced by contact electrification. In the method, the uniform electric field is applied to a droplet suspended in dielectric oil. The horizontal movement of a droplet is determined by the balance between electric and drag force. The drag force exerted on a droplet has been calculated by Hadamard-Rybczinski solution. The effects of a droplet size, electrolyte concentration of an aqueous droplet on the amount of charge have been examined.

Tuesday, November 22, 2011 3:05PM - 5:15PM –

Session S1 Geophysical Flows: General III 301

3:05PM S1.00001 Large-eddy simulation of flow past a real-life stream restoration structure[1]

SEOKKOO KANG, FOTIS SOTIROPOULOS, University of Minnesota — We carry out high-resolution large-eddy simulation (LES) of flow around a rock vane, which is a widely used stream restoration structure. Mean velocities and turbulence statistics collected downstream of the rock vane installed in a laboratory flume are compared with the LES results. The comparisons demonstrate that the LES is able to accurately predict the measured mean velocities and turbulence statistics. The simulation shows that the rock vane effectively directs the oncoming flow away from the structure and creates a reduced velocity region in the downstream region. The computed results also reveal that the rock vane creates strong secondary helical flow that directs the near-bed flow toward the sidewall to which the rock vane is attached. This finding points to the conclusion that the downstream secondary flow can create deposition of sediments near the sidewall in a mobile bed condition, which can serve as an important mechanism for protecting near-bank scour in natural streams.

[1]This work was supported by National Center for Earth-surface Dynamics (NCED), ECORIVER21 project in South Korea, National Cooperative Highway Research Program (NCHRP) and Minnesota Supercomputing Institue (MSI).

3:18PM S1.00002 Numerical Simulation of turbulent flow and sediment transport around real-life stream restoration structures[1] , ALI KHOSRONEJAD, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, University of Minnesota — Local scour around three real-life in-stream restoration rock structures, including a rock-vane, a cross-vane, and a J-hook, is investigated numerically. To overcome the difficulties of generating high quality boundary-fitted meshes in natural mobile-bed channels with embedded rock structures, we employ the fluid-structure interaction Curvilinear Immersed Boundary (CURVIB) method adapted for morphodynamic simulations (Khosronejad et al., Adv. Water Res., 34(7) 2011). The mobile bed and the immersed structures are discretized with an unstructured triangular mesh and are treated as sharp-interface immersed boundaries embedded in a background curvilinear mesh used to discretize the fluid domain. The flow field is simulated by solving the unsteady RANS equations closed with the k-w turbulence model. The bed evolution is calculated by solving the Exner equation using an unstructured, finite-volume formulation. Comparisons with measurements show that the computed results capture both the spatial and temporal features of scour and deposition patterns with good accuracy.

[1]This work was supported by NSF Grants EAR-0120914 and EAR-0738726, National Cooperative Highway Research Program Grant NCHRP-HR 24-33, and a Grant from Yonsei University.

3:31PM S1.00003 RANS Simulation of Passive Scalar Residence Times and Exchange Processes in Idealized and Natural Stream Systems[1] , KEVIN DROST, TRACIE JACKSON, ROY HAGGERTY, SOURABH APTE, Oregon State University — Natural stream systems contain a variety of dead zones characterized by flow separation, a mixing layer, and a recirculation zone. These dead zones play an important role in stream solute transport studies. Previous published work has focused on idealized storage zone geometries studied in laboratory flumes. Using RANS simulations, this study first examines these idealized geometries to determine the appropriate scaling relationships between idealized dead zone geometries and the residence times of a passive scalar. These scaling relationships are then applied to measurements from natural systems. The field-measured geometries are located in Oak and Soap creeks near Corvallis, Oregon. Field measurements for the natural systems included: (a) survey measurements to delineate storage zone morphologies; (b) Marsh-McBirney and acoustic Doppler velocimetry measurements for model boundary conditions and computation of turbulence parameters; and (c) continuous salt injections within storage zones and electrical conductivity measurements at point locations in the main channel and storage zones to quantify exchange rates and residence times.

[1]This work is sponsored by NSF-EAR project #0943570.

3:44PM S1.00004 Shallow-water models for gravity currents/intrusions in double-continuous-stratification systems , MARIUS UNGARISH, Technion Haifa Israel — High-Reynolds-number Boussinesq gravity currents and intrusions systems, in which both the ambient and the propagating "current" are linearly stratified, are considered. The main focus is on a current of fixed volume released from a lock; the height ratio of the fluids H, and the stratification parameter of the ambient S, are quite general. It is shown that a one-layer shallow-water model, in which the internal stratification enters as a new dimensionless parameter, $\sigma \in [0, 1]$, provides insightful results. In general, the speed of propagation decreases when the internal stratification becomes more pronounced (σ increases), and in some cases the current runs out of driving force. This model is a versatile and robust self-contained prediction tool, which can be applied to both rectangular and axisymmetric geometries, and to quite general continuously- stratified systems.

3:57PM S1.00005 Transitions of the propagation phases for non-Newtonian gravity currents , MIJANUR CHOWDHURY, FIRAT TESTIK, Civil Engineering Department, Clemson University, Clemson, SC, USA — Transitions of the propagation phases for both two-dimensional and axisymmetric non-Newtonian gravity currents were investigated experimentally and theoretically. Fluid mud gravity currents, which exhibit power-law (shear thinning) rheological properties, were generated for constant-volume (in a flume) and constant-flux (in a flume and a three-dimensional tank) release configurations. Experimental observations indicated that, similar to their Newtonian counterparts (e.g. saline gravity currents), fluid mud gravity currents exhibit inertial and viscous propagation phases, preceded by either slumping (for the case of constant-volume release) or chaotic jet (for the case of constant-flux release) phase. When the currents make transitions from inertial to viscous phase, a thickening-thinning behavior was observed. Order-of-magnitude expressions for the transition time and position were derived and predictions of these expressions were compared to the experimental observations. A Moody-like diagram based upon a new friction factor and Reynolds number for power-law gravity currents is proposed to identify the transition from the inertial to viscous propagation phase.

4:10PM S1.00006 Turbulent Entrainment into Non-Newtonian Fluid Mud Gravity Currents , MICHAEL JACOBSON, FIRAT TESTIK[1], Clemson University — This study presents insights into turbulent entrainment of ambient water into fluid mud gravity currents. It is well established that fluid mud suspensions exhibit pseudo-plastic behavior. Gravity current laboratory experiments were conducted for constant-volume release configuration with different initial concentrations of fluid mud, representing different rheological properties (i.e. different Power-law model constants). A high quality data set of concentration and velocity profiles of fluid mud gravity currents was collected to calculate the entrainment velocity, w_e. The entrainment ratio ($E = w_e/U, U$ – characteristic velocity) was calculated following the well-accepted Morton-Taylor-Turner entrainment hypothesis, which states that the inflow across the edge of a turbulent flow is proportional to some characteristic velocity. The entrainment ratio was further measured qualitatively using a light opaqueness technique. A semi-empirical parameterization for the entrainment ratio is proposed. The findings of this study are expected to be of significance for modeling various non-Newtonian gravity currents, in particular for modeling fluid mud gravity currents generated during dredge disposal operations in coastal waters.

[1]Contact Author

4:23PM S1.00007 Non-Boussinesq internal bores: Bridging the gap between the single layer, and boussinesq cases , ZACHARY BORDEN, University of California, Santa Barbara, TILMAN KOBLITZ, Technical University of Denmark, ECKART MEIBURG, University of California, 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. Our previous study used 2D direct numerical simulations in order to directly examine the energy fluxes within a bore, and was able to provide us with an appropriate closure relation in the Boussinesq case. Now, we have extended our simulations to non-Boussinesq bores in order to generate an analytical model that bridges the gap between single, and two-layer models. We also perform 3D large eddy simulations to confirm that our model and results generalize to three dimensions, and higher Reynolds numbers.

4:36PM S1.00008 ABSTRACT WITHDRAWN —

4:49PM S1.00009 Radiative instability of a strong anticyclonic vortex in a rotating and stratified fluid , JUNHO PARK, PAUL BILLANT, LadHyX, Ecole Polytechnique — In strongly stratified fluids, an axisymmetric vertical columnar vortex with an angular velocity Ω is unstable because of a spontaneous radiation of internal waves. This radiative instability tends to be stabilized in the presence of a cyclonic background rotation Ω_b so that it is generally expected to not affect large-scale vortices in geophysical flows. In contrast, we show that an anticyclonic vortex with low negative Rossby number $Ro = \frac{\Omega}{\Omega_b} \geq -1$, which is centrifugally stable, can be also unstable due to radiative instability when the azimuthal wavenumber m is sufficiently large: $|m| \geq 3$. This new radiative instability has been investigated for several vortex profiles and we have found that it occurs if the potential vorticity of the vortex has a sufficiently steep radial profile. The properties of this instability and its mechanism will be explained by an asymptotic analysis for large azimuthal wavenumber. Moreover, its importance for geophysical anticyclonic vortices such as meddies will be discussed.

5:02PM S1.00010 Instabilities in sediment-laden systems , PETER BURNS, ECKART MEIBURG, UCSB — When a layer of particle-laden fresh water is placed above clear, saline water, both Rayleigh-Taylor and double-diffusive instabilities may arise. In the absence of salinity, the dominant parameter is the ratio of the particle settling velocity to the viscous velocity scale. As long as this ratio is small, particle settling has a negligible influence on the instability growth. However, when the particles settle more rapidly than the instability grows, the growth rate decreases inversely proportional to the settling velocity. In the presence of a stably stratified salinity field, this picture changes dramatically. An important new parameter is the ratio of the height of the nose region that contains both salt and particles to the thickness of the salinity interface. If this ratio is small (large) the dominant instability mechanism will be double-diffusive (Rayleigh-Taylor) dominant. In contrast to situations without salinity, particle settling can have a destabilizing effect and significantly increase the growth rate. Scaling laws obtained from the linear stability results are seen to be consistent with experimental observations and theoretical arguments put forward by other authors.

Tuesday, November 22, 2011 3:05PM - 5:02PM –
Session S4 Drops XI: Interaction with Superhydrophobic Surfaces 307

3:05PM S4.00001 Liquid-encapsulating surfaces: overcoming the limitations of superhydrophobic surfaces for robust non-wetting and anti-icing surfaces , J. DAVID SMITH, Massachusetts Institute of Technology, RAJEEV DHIMAN, MIT, KRIPA VARANASI[1], Massachusetts Institute of Technology — In this work we address fundamental limitations of superhydrophobic surfaces for non-wetting and anti-icing applications by impregnating them with a hydrophobic liquid. The encapsulated liquid serves as a barrier to the penetration of impinging water droplets and forces preferential condensation and frost formation on texture tops. We conducted droplet impact and roll-off experiments to assess the robustness of liquid-encapsulating micro- and nano-scale textured surfaces and found that their ability to shed droplets was improved dramatically. Furthermore, environmental scanning electron microscope experiments demonstrated that frost formation as well as condensation occurs preferentially on these surfaces thereby limiting ice contact to texture tops only. Ice adhesion strength was quantified using a custom-built adhesion testing apparatus to demonstrate greatly enhanced anti-icing performance of the liquid-encapsulating surfaces compared to superhydrophobic surfaces.

[1]Assistant Professor of Mechanical Engineering

3:18PM S4.00002 Two dimensional microfluidic devices for sorting, mixing, and analyzing drops using superhydrophobic surfaces , MICHAEL NILSSON, HRISTO GOUMNEROV, JONATHAN ROTHSTEIN, University of Massachusetts Amherst — The effect of sharp transitions in contact angle hysteresis of a superhydrophobic surface on water droplet motion is investigated. The surfaces are created by sanding naturally hydrophobic Teflon, which results in a surface possessing microscale roughness. With careful sanding and masking of the surface, regions with similar advancing contact angles near 150 degrees but varying contact angle hysteresis can be created. This talk presents detailed results on how these sharp transitions can deflect drops when the transition is at an angle to the droplet trajectory. The physics of the droplet-transition interaction is discussed and the sensitivity of the alignment of the angle of transition and Weber number is presented. Deflection using an angled transition in contact angle hysteresis is selective, demonstrating the ability of this fabrication method to sort drops based on size, speed, and wettability. Results of surfaces with stripes of different contact angle hysteresis are presented, showing the ability to more effectively deflect. Finally, a two-dimensional droplet reactor is presented that encompasses droplet sorting, coalescence, mixing, and trapping of two droplets.

3:31PM S4.00003 Hierarchical roughness of sticky and non-sticky superhydrophobic surfaces , MUHAMMAD AKRAM RAZA, STEFAN KOOIJ, AREND VAN SILFHOUT, HAROLD ZANDVLIET, BENE POELSEMA, University of Twente — The importance of superhydrophobic substrates (contact angle >150° with sliding angle <10°) in modern technology is undeniable. We present a simple colloidal route to manufacture superstructured arrays with single- and multi-length-scaled roughness to obtain sticky and non-sticky superhydrophobic surfaces. The largest length scale is provided by (multi-)layers of silica spheres (1μm, 500nm and 150nm diameter). Decoration with gold nanoparticles (14nm, 26nm and 47nm) gives rise to a second length scale. To lower the surface energy, gold nanoparticles are functionalized with dodecanethiol and the silica spheres by perfluorooctyltriethoxysilane. The morphology was examined by helium ion microscopy (HIM), while wettability measurements were performed by using the sessile drop method. We conclude that wettability can be controlled by changing the surface chemistry and/or length scales of the structures. To achieve truly non-sticky superhydrophobic surfaces, hierarchical roughness plays a vital role.

3:44PM S4.00004 Reducing contact time of drops on superhydrophobic surfaces , JAMES BIRD, RAJEEV DHIMAN, HYUK-MIN KWON, KRIPA VARANASI, MIT — When water drops impact on to a superhydrophobic surface, the drops can recoil to such an extent that they completely bounce off the solid material. The time it takes for the drop to spread and recoil – the contact time – scales with the hydrodynamic inertial-capillary timescale. However, there is evidence that the coefficient of this scaling depends on surface-structure interactions, such as pinning. Here we investigate how surface interactions can influence droplet contact time, and we compare our results to existing models. We highlight an assumption in the current theory that imposes a lower-bound on the contact time. By designing around this constraint, we demonstrate novel superhydrophobic surfaces on which water droplets impact with shorter contact times than previously thought possible.

3:57PM S4.00005 Fog interaction with a textured hydrophobic surface: deposition, growth, and wetting , TONY S. YU, Brown University, JOONSIK PARK, HYUNEUI LIM, Korea Institute of Machinery and Materials, KENNETH S. BREUER, Brown University — A hydrophobic surface patterned with micro-pillars can exhibit "superhydrophobic" behavior, wherein drops only touch the tops of pillars (the so-called Cassie-Baxter state). This state, however, requires drops much larger than the pillar size/spacing. In contrast, drops smaller than the microtexture can adsorb both on top of and in- between these pillars and readily stick to the solid. In this study, we image the deposition of drops from fog (i.e. micron-sized droplets suspended in air) flowing over a substrate textured with square pillars, which are tens-of-micron in size. After texturing, the surface is coated with a hydrophobic monolayer. While drop growth at early time is dominated by deposition, late-time growth is dominated by coarsening due to the coalescence of adjacent drops. In this talk, we quantify the evolution of the size of these adsorbed drops. In addition, we show how closely-spaced pillars limit the size distribution of growing drops and enhance their spatial ordering.

4:10PM S4.00006 Multiscale computation of wetting transitions on micro/nanoscale solid structures , CARLOS COLOSQUI, Princeton University, MICHAIL KAVOUSANAKIS, ATHANASIOS PAPATHANASIOU, National Technical University of Athens, IOANNIS KEVREKIDIS, Princeton University — Modern microfabrication techniques allow the construction of solid surfaces with micro or nanoscale structure. The artificial micro/nanostructure, when properly designed, produces smart surfaces capable of exhibiting surperhydrophobic or superhydrophilic wetting properties in an addressable manner The design and control of transitions between superhydrophobic and superhydrophilic states using minimal external actuation (e.g. via electro-wetting, UV radiation, etc.) is an active research area with enormous impact in microfluidics and materials science. In this work, we demonstrate a computational methodology that allows microscopic or mesoscopic models to perform numerical continuation and stability analysis at macroscopic level. The method's ability to compute stable and unstable wetting states via accurate dynamic simulations is extremely valuable for the optimal engineering design of micro/nanostructures. The computational methodology is demonstrated by employing a novel lattice Boltzmann model that captures critical interfacial phenomena such as disjoining pressure and vapor adsorption at the solid-fluid surface.

4:23PM S4.00007 3D aspects of droplet coalescence during dropwise condensation on superhydrophobic surfaces , KONRAD RYKACZEWSKI, JOHN HENRY J. SCOTT, SUKUMAR RAJAURIA, NIST, JEFF CHINN, AMY M. CHINN, WANDA JONES, Integrated Surface Technologies, Inc — Only a few selected natural and artificial surfaces with water contact angles above 150 degrees retain their superhydrophobic characteristics during water condensation. On these robust superhydrophobic surfaces, individual droplets emerge out of a few micrometer wetted area between the nanostructures and initially grow mainly by increasing their contact angle. After reaching a nearly spherical shape with a diameter between 4 um and 6 um, the droplets grow in a near constant contact angle mode while remaining in an immobile Wenzel state. Microdroplets can depart the surface by coalescing with another large drop and forming a new drop in the mobile Cassie-Baxter state. Here we report that high contact angle primary drops can project over growing nano-to-microscale satellite droplets. We show that the large primary drops can sweep up the small satellite droplets without wetting their nucleation site, promoting rapid condensation of multiple satellite droplets from the same nucleation site. We discuss the effect of this coalescence mechanism on the drop size distribution and heat transfer during the dropwise condensation process.

4:36PM S4.00008 Impact of Molten Metal Droplets on Textured Surfaces , RAJEEV DHIMAN, KRIPA VARANASI, MIT — Most studies of droplet impact on micro/nano-textured surfaces have been done with water droplets. In this work, we studied impingement of molten metal droplets on micro and nanotextured surfaces which is relevant to many industrial applications such as thermal spray coating, spray forming, and solder jet bumping. The phenomenon is complicated by the fact that droplet spreading and freezing occur simultaneously. We used molten tin droplets (2.7 mm diameter) and deposited them on 10 um square posts surfaces made of silicon by standard photolithography. The impact process was photographed with a high-speed video camera and droplet deformations were analyzed. Surface temperature (25-240° C), impact velocity (0.5-3 m/s), and micropost spacing (5-50 μm) were the key parameters varied. We found that the maximum spreading diameter of the droplet increased with surface temperature and impact velocity, which could be predicted with the help of a simple mathematical model quite well. Droplets splashed through formation of satellite droplets at higher velocities.

4:49PM S4.00009 Numerical simulations of drop impact on superhydrophobic structured surfaces[1] , DAVIDE GUZZETTI, STEFANO LARENTIS, NICOLA PUGNO — During the last decade drop impact dynamics on superhydrophobic surfaces has been intensively investigated because of the incredible properties of water repellency exhibited by this kind of surfaces, mostly inspired by biological examples such as Lotus leave. Thanks to the recent progress in micro-fabrication technology is possible to tailor surfaces wettability defining specific pillar-like structured surfaces. In this work, the behavior of impinging drops on these pillar-like surfaces is simulated, characterizing temporal evolution of droplets contact radius and drop maximal deformation dependence on Weber number. Numerical simulations results are compared with theoretical and experimental results guaranteeing simulation reliability. Fingering patterns obtained from drop impact has been studied obtaining a correlation between number of fingers and Weber number. Drop fragmentation pattern obtained from simulations supports the proposed correlation. Different drop impact outcomes (e.g. rebound, fragmentation) on structured superhydrophobic surfaces are simulated, focusing on the influence of micro-structured surface geometrical pattern. This investigation is relevant in order to define design rules for possible reliable non wettable surfaces.

[1] Financial support by Alta Scuola Politecnica

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S5 Drops XII: Coalescence 308

3:05PM S5.00001 Measurements of liquid film thickness for a droplet at a two-fluid interface , GOSSE OLDENZIEL, FOM, Delft University of Technology, JERRY WESTERWEEL, RENE DELFOS, Delft University of Technology — Coalescence of a droplet at a two-fluid interface is studied at Bond numbers larger than one and at three different values of the viscosity ratio. Both the thickness of the liquid film between the rising droplet and the two-fluid interface, and the location of film rupture are measured using Laser Induced Fluorescence. Particle Image Velocimetry was applied to the flow in the film. It is found that the film thins asymetrically, and that the time interval between collision and film rupture is shorter than predicted by commonly used models. The film ruptures at an off-center location. It can be concluded that asymmetric film drainage speeds up coalescence.

3:18PM S5.00002 Coalescence of liquid drops , SANTOSH APPATHURAI, MICHAEL HARRIS, OSMAN BASARAN, Purdue University, JOSEPH PAULSEN, JUSTON BURTON, SIDNEY NAGEL, University of Chicago — Drop coalescence plays a central role in industrial contexts, e.g. emulsions, sintering processes, and inkjet printing, as well as in everyday phenomena ranging from dripping faucets to raindrops in clouds. During coalescence, two drops touch each other and then merge as a liquid bridge grows from microscopic scales to a size comparable to the drop diameter. This process has been thought to have just two regimes: a highly viscous one during the initial stages, pulling the drops together, and an inertial one later on, dominated by interface deformations near the neck. We use high-speed imaging, electrical measurements and full Navier-Stokes simulations to reveal a new regime that governs the asymptotic dynamics of coalescence for any finite viscosity in three dimensions.

3:31PM S5.00003 Effects of Marangoni Stresses on Drop Coalescence , SOFYA KABACHEK, H. PIROUZ KAVEHPOUR, UCLA — In many applications involving drop-drop interactions, such as alcohol-based fuel cells and many microfluid devices, miscible fluids are not necessarily of the same kind. To better understand these phenomena, further insight into coalescence of different fluids is essential. When miscible drops of different liquids come in contact, coalescence often occurs and a surface tension difference may arise at the interface. This difference plays an important role in the process due to the Marangoni effect. To quantify the extent of this effect we conducted a parametric study on the effects of surface tension variation on drop coalescence (partial and full); in particular coalescence time and drop diameter ratios were investigated. A high speed digital camera with high resolution was utilized to study the evolution of drops during the coalescence process. We showed that coalescing drops with higher surface tension than the reservoir fluid, behave similarly to drops coalescing in a viscous environment. A new physical model was developed in good agreement with the experimental data.

3:44PM S5.00004 Mixing in Sessile Drops Merging on a Surface , SHELLEY ANNA, YING ZHANG, SAMUEL OBERDICK, STEPHEN GAROFF, Carnegie Mellon University — We investigate the mixing of two sessile drops that merge on a surface. The drops consist of low viscosity glycerol-water mixtures deposited on a silicone elastomer surface with contact angle near 90°. We observe the shape of the drops and the location of their intersection by placing a fluorescent dye in one drop and using a laser light sheet to image a plane perpendicular to the surface. The initial healing of the meniscus bridge between the merging drops, and the damping of capillary waves appearing on their surfaces occur on timescales comparable to the inertio-capillary relaxation time. However, the interface between the two fluids remains sharp, broadening diffusively over several minutes. The shape of the merged drops and the boundary between them also continues to evolve on a timescale of minutes. This later motion is controlled by gravity, capillary pressure, and viscous stresses. Images of the 3D drop shape indicate that small contact line motions are correlated to the slow relaxation. Although the two drops contain identical liquids except for the presence of the dye, the shape of the interface consistently evolves asymmetrically, assuming a characteristic crescent shape. We note that very tiny surface tension gradients can produce an asymmetric flow like the one observed here. We characterize the long timescale flow as a function of the drop sizes, and we use numerical simulations to aid in elucidating the essential physics.

3:57PM S5.00005 Vortex-driven charge transfer between partially coalescing droplets , WILLIAM RISTENPART, J.C. CREASEY, B.S. HAMLIN, University of California at Davis — Oppositely charged drops fail to coalesce above a critical field strength, despite the attractive force between the opposite charges. Here we report the existence of a critical ionic conductivity below which oppositely charged drops only partially coalesce. The degree of coalescence of water drops in oil can be tuned from complete coalescence at low field strengths to complete non-coalescence at high field strengths, thus providing external control over the size of the resulting daughter droplet. Strikingly, in this regime the size and charge of the daughter droplet are both independent of the conductivity. We present evidence suggesting the charge transfer is instead dominated by convection associated with the capillary-driven penetration of a vortex into the larger drop, and we demonstrate that the size of the daughter droplet is consistent with a scaling model based on a balance between capillary-driven inertia and electrostatic repulsion.

4:10PM S5.00006 ABSTRACT WITHDRAWN —

4:23PM S5.00007 Drop impacting on a hydrophobic elastic beam[1] , DANIEL CHIQUE CANACHE, SUNGHWAN JUNG, Department of Engineering Science and Mechanics, Virginia Tech — Plant suraces found in nature often exhibit hydrophobic wetting properties; in particular, the surface of leaves are an example. When a water drop impacts a leaf a unique system of coupled solid and fluid mechanics is observed. By replacing the leaf as as simple thin polycarbonate cantilever beam it is possible to create a workable model for the system. A high-speed camera allows detailed observation of the dynamics of the beam and drop at the moment of impact. Through image analysis, the position and shape of the beam and drop are analyzed to calculate bending energy and kinetic energy. Experiments show that the available energy in the system is close to 0.1 mJ. The results of this experiment provide insight into energy harvesting from raindrops using a piezo cantilever.

[1]COE Diversity Summer Research Program

4:36PM S5.00008 Drop interaction with the ejecta sheet , M.-J. THORAVAL, S.T. THORODDSEN, King Abdullah University of Science and Technology, Thuwal 23955- 6900, Saudi Arabia, K. TAKEHARA, T.G. ETOH, Kinki University, Osaka, Japan, P. RAY, C. JOSSERAND, S. ZALESKI, Institut D'Alembert, Paris, France — We studied experimentally and numerically the dynamics of the ejecta sheet produced by a drop impacting on a deep pool of the same liquid at high Reynolds and Weber numbers. Ultra-high speed imaging revealed a diversity of evolutions by using different mixtures of water and glycerine, and different impact velocities. We observed a transition from a smooth ejecta sheet to a more irregular splashing for a Reynolds number of 3500. In this transition regime, the ejecta sheet interacts with the impacting drop. This interaction can pull the ejecta sheet towards the centre of the drop at lower splash parameters, or generate a bumping on the ejecta sheet moving outwards at higher splash parameters. The volume of fluid (VOF) code Gerris was used to reproduce numerically this peculiar dynamics in axisymmetric conditions. Very good agreement with the experiments was reached by using adaptive refinement and parallelization of the calculations.

4:49PM S5.00009 Dimensional crossover in the viscous dynamics of coalescence of liquid drops confined in between two plates[1] , MARIA YOKOTA, KO OKUMURA, Department of Physics, Graduate School of Ochanomizu University — Recently, it has been established that the dynamics of coalescence of liquid drops, driven by capillary force, is balanced by viscous force at shorter times (or in viscous drops) and by inertial force at longer times (or in less-viscous drops) [1, 2]. This has been confirmed also in two-dimensional coalescence [3]. Here, we study coalescence of a liquid droplet to a bath of the same liquid in a confined geometry of a Hele-Shaw cell [4]. We followed the dynamics of the neck which bridges the drop and the liquid bath. We find, in a single coalescence event, a crossover from a three dimensional viscous dynamics to another quasi two-dimensional viscous dynamics. This crossover is established by demonstrating clear data collapse, thanks to simple dimensional arguments. We discuss further the third scaling regime between the two viscous regimes, together with an unusual charge effect on the coalescence which makes the dynamics self-similar.

[1] Eggers J, Lister JR, Stone HA (1999) J. Fluid. Mech. 401:293-310.
[2] Aarts DGAL, Lekkerkerker HNW, Guo H, Wegdam GH, Bonn D (2005) Phys. Rev. Lett. 95:164503.
[3] Burton JC, Taborek P (2007) Phys. Rev. Lett. 98:224502.
[4] Yokota M, Okumura K (2011) Proc. Nat. Acad. Sci. 108:6395.

[1]This work is supported by the JSPS.

5:02PM S5.00010 The Effect of Surfactants on the Breakup of an Axisymmetric Laminar Jet , JUSTIN WALKER, RICHARD CALABRESE, University of Maryland — The breakup of a laminar axisymmetric jet is a well-studied fluid dynamics phenomenon, first studied by Savart (1833) and Rayleigh (1879). Many papers have been published over the years describing the theory of jet breakup, such as the paper by Tomotika (1935). More recently, many studies have been performed using various computational simulations to better understand the mechanics of jet breakup, notable among these are Homma et al. (2006). Despite the extensive literature on the topic, the impact of surface active agents on jet breakup has received limited attention, whether due to the system's inherent complexity or a poor understanding of the mechanics of the action of surface active agents themselves. In this study, the drop size distribution and jet breakup length resulting from the breakup of liquid jet systems were studied experimentally. Jets were formed by forcing a fluid through a narrow capillary using pneumatic pressure. Experiments involving oil-water jets with aqueous surfactants were performed. Several distinct regimes were identified based on hydrodynamic and physicochemical conditions. Jet length was found to increase with surfactant concentration, while droplet diameter was found to decrease (dependent on jet regime). A Semiempirical model to predict the breakup length of Jets in the presence of surfactants is also proposed.

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S6 Turbulence: Mixing III 309

3:05PM S6.00001 Schmidt-number dependence in turbulent mixing: very low Schmidt numbers and spectral transfer[1] , P.K. YEUNG, Georgia Tech, K.R. SREENIVASAN, New York Univ, K.P. IYER, D. BUARIA, Georgia Tech — The physics of turbulent mixing depends on both the Reynolds number and Schmidt number (Sc), which varies widely in applications and leads to different scaling regimes. The case of $Sc \ll 1$, which is relevant in liquid metals and astrophysics, is perhaps the least understood since laboratory data are difficult to obtain. We have performed direct numerical simulations of passive scalars of Sc from 1/32 to 1/512, on a periodic domain of larger size than usual to accommodate the growth of large scales in the scalar fields, and with a very small time step to resolve the time scales of molecular diffusion. For $Sc = 1/128$ and 1/512 the spectrum obtained appears to support $k^{-17/3}$ inertial-diffusive behavior proposed by Batchelor, Howells & Townsend (1959) although results at higher Reynolds numbers are required. Calculations of spectral transfer, including the transfer flux, indicate that the spectral cascade is greatly suppressed, which implies a number of classical notions such as dissipative anomaly and local isotropy become inapplicable in this regime. Together with other recently published data the new results also enable progress towards a unified view of Schmidt number dependence for small-scale turbulent mixing.

[1]NSF Grants CBET-1139037

3:18PM S6.00002 A conditional sampling-based method for noise and resolution corrections for scalar dissipation rate measurements[1] , CHENNING TONG, JIAN CAI, Clemson University — A conditional sampling-based method for correcting noise and resolution effects for scalar dissipation rate measurements is developed. Noise and resolution effects on the measured dissipation rate have opposite trends, making their separation and accurate corrections difficult. A major task in dissipation rate correction, therefore, is to isolate each effect. The method presented in this work uses instantaneous local scalar mean and variance as conditioning variables, and is based in part on Kolmogorov's refined similarity hypotheses. It ensures selection of instantaneous fully resolved local scalar fields, which are analyzed to determine the measurement noise. Noise correction is applied to potentially under-resolved local scalar fields, also selected using the conditional-sampling procedure, effectively separating the effects of noise from those of resolution. The error function is used as a model for the potentially under-resolved local scalar fields to evaluate their dissipation length scales and to make corrections for the dissipation rate. The present method uses local instead of spectral analyses; therefore, can be applied to the mean scalar dissipation rate conditional on the scalar values. An application of the method to temperature dissipation rate in a slightly heated turbulent jet shows excellent results, validating the method. The method has also been applied to turbulent flames.

[1]Supported by NSF and AFOSR.

3:31PM S6.00003 Can fractal objects operate as efficient inline mixers?[1] , SYLVAIN LAIZET, JOHN CHRISTOS VASSILICOS, Imperial College London, TURBULENCE, MIXING AND FLOW CONTROL GROUP TEAM — Recently, Hurst & Vassilicos, PoF 2007, Seoud & Vassilicos, PoF 2007, Mazellier & Vassilicos, PoF, 2010 used different multiscale grids to generate turbulence in a wind tunnel and have shown that complex multiscale boundary/initial conditions can drastically influence the behaviour of a turbulent flow, but that the detailled specific nature of the multiscale geometry matters too. Multiscale (fractal) objects can be designed to be immersed in any fluid flow where there is a need to control and design the turbulence generated by the object. Different types of multiscale objects can be designed as different types of energy-efficient mixers with varying degrees of high turbulent intensities, small pressure drop and downstream distance from the grid where the turbulence is most vigorous. Here, we present a 3D DNS study of the stirring and mixing of a passive scalar by turbulence generated with either a fractal square grid or a regular grid in the presence of a mean scalar gradient. The results show that: (1) there is a linear increase for the passive scalar variance for both grids, (2) the passive scalar variance is ten times bigger for the fractal grid, (3) the passive scalar flux is constant after the production region for both grids, (4) the passive scalar flux is enhanced by an order of magnitude for the fractal grid.

[1]We acknowledge support from EPSRC, UK.

3:44PM S6.00004 Instabilities and mixing in a quasi-2D Lorentz-force driven flow , RADFORD MITCHELL, ROMAN GRIGORIEV, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology — In this talk we describe the transition to weak turbulence in a Kolmogorov flow inside a thin layer of electrolyte driven by a steady current in experiment and a computational model. We find the mixing efficiency of the flow to be a non-monotonic function of the driving current, mirroring the temporal complexity of the flow pattern. We also illustrate the generic mechanism of mixing by time-periodic two-dimensional flows near a supercritical Hopf bifurcation and provide a perturbative description of the mixing process.

3:57PM S6.00005 Toward topology-based characterization of small-scale mixing in compressible turbulence , SAWAN SUMAN, SHARATH GIRIMAJI, Texas A&M University — Turbulent mixing rate at small scales of motion (molecular mixing) is governed by the steepness of the scalar-gradient field which in turn is dependent upon the prevailing velocity gradients. Thus motivated, we propose a velocity-gradient topology-based approach for characterizing small-scale mixing in compressible turbulence. We define a mixing efficiency metric that is dependent upon the topology of the solenoidal and dilatational deformation rates of a fluid element. The mixing characteristics of solenoidal and dilatational velocity fluctuations are clearly delineated. We validate this new approach by employing mixing data from direct numerical simulations (DNS) of compressible decaying turbulence with passive scalar. For each velocity-gradient topology, we compare the mixing efficiency predicted by the topology-based model with the corresponding conditional scalar variance obtained from DNS. The new mixing metric accurately distinguishes good and poor mixing topologies and indeed reasonably captures the numerical values. The results clearly demonstrate the viability of the proposed approach for characterizing and predicting mixing in compressible flows.

4:10PM S6.00006 Influence of Forcing Structure on Two-Dimensional Weak Turbulence , YANG LIAO, DOUGLAS KELLEY, NICHOLAS OUELLETTE, Yale University — The dependence of the dynamics of two-dimensional turbulence on its forcing topology is an interesting and practical question for experiments. We generate quasi-two-dimensional flows in thin layers of salt water with a forcing geometry that is either a lattice of alternating vortices or an array of alternating shear bands. We observe that the vortex flow has more fluctuating energy, but that the shear flow has larger spatial gradients. We attribute these differences to the fact that shear forcing imposes fewer constraints on the flow and allows the production of smaller length scales.

4:23PM S6.00007 Cloud microphysical effects of turbulent mixing and entrainment , RAYMOND SHAW, Michigan Technological University, BIPIN KUMAR, JOERG SCHUMACHER, Ilmenau University of Technology — Turbulent mixing and entrainment at the boundary of a cloud is studied by means of direct numerical simulations that couple the Eulerian description of the turbulent velocity and water vapor fields with a Lagrangian ensemble of cloud water droplets that can grow and shrink by condensation and evaporation. The focus is on detailed analysis of the relaxation process of the droplet ensemble during the entrainment of subsaturated air, in particular the dependence on turbulence time scales, droplet number density, initial droplet radius and particle inertia. We find that the droplet evolution during the entrainment process is captured best by a phase relaxation time that is based on the droplet number density with respect to the entire simulation domain and the initial droplet radius. Even under conditions favoring homogeneous mixing, the probability density function of supersaturation at droplet locations exhibits strong negative skewness, consistent with droplets near the cloud boundary being suddenly mixed into clear air, but rapidly approaches a narrower, bimodal shape. The droplet size distribution, which is initialized as perfectly monodisperse, broadens and also becomes somewhat negatively skewed. However, particle inertia and gravitational settling impose a sharp cutoff in both tails of the droplet size distribution during later stages of the mixing.

4:36PM S6.00008 Rapid growth of cloud droplets by turbulence

JOHN CHRISTOS VASSILICOS, VASSILIS DALLAS, Imperial College London — Assuming perfect collision efficiency, we demonstrate that turbulence can initiate and sustain rapid growth of very small water droplets in air even when these droplets are too small to cluster, and even without having to take gravity and small-scale intermittency into account. This is because the range of local Stokes numbers of identical droplets in the turbulent flow field is broad enough even when small-scale intermittency is neglected. This demonstration is given for turbulence which is one order of magnitude less intense than typically in warm clouds but with a volume fraction which, even though small, is nevertheless large enough for an estimated a priori frequency of collisions to be ten times larger than in warm clouds. However, the time of growth in these conditions turns out to be one order of magnitude smaller than in warm clouds.

4:49PM S6.00009 The emergence of correlations from distant sources in turbulent flows

MIHKEL KREE, IRPHE, Aix Marseille Universite, JÉRÔME DUPLAT, Universite Joseph Fourier, Grenoble, EMMANUEL VILLERMAUX, Aix Marseille Universite — A scalar field injected from a point source in a turbulent flow resolves into a set of elongated sheet-like structures whose concentration probability distribution $P(C)$ is well understood in terms of the interplay between stretching histories of fluid particles, and molecular diffusion. Additionally, neighboring sheets may diffusively merge and evolve further in an undistinguishable fashion. We report here on experiments using two distinct dyes (Fluorescein and Rhodamine in water) injected form sources separated by a variable distance s in a turbulent jet. This method allows to separate the total concentration field C into its subparts C_1 and C_2 originating from each sources. Joint probability distributions $Q(C_1, C_2)$ and correlations between the fields can thus be measured. We evidence this way the crossover scale above which the fields are and remain independent in the flow.

5:02PM S6.00010 Spatiotemporal persistence of spectral fluxes in two-dimensional weak turbulence[1]

DOUGLAS H. KELLEY, NICHOLAS T. OUELLETTE, Yale University — Nonlinearity in a dynamical system necessarily leads to coupling between different spatial and temporal frequencies. Using a recently developed filtering technique, we study the spatiotemporal properties of the scale-to-scale fluxes of energy and enstrophy in a weakly turbulent experimental quasi-two-dimensional flow. Although these spectral properties vary in time and space, we show that they persist along the Lagrangian trajectories of fluid elements for times that can be nearly as long as the correlation time of the velocity field itself. Additionally, we show that at small scales, the spectral energy flux persists longest for fluid elements in strongly hyperbolic regions of the flow, whereas at large scales it persists in strongly elliptic regions.

[1]This work was supported by the U.S. National Science Foundation under grant no. DMR-0906245.

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S7 Tubulent Boundary Layers IX 310

3:05PM S7.00001 An experiment to determine the frequency response of hot-wire anemometers to velocity fluctuations

NICHOLAS HUTCHINS, JASON MONTY, IVAN MARUSIC, University of Melbourne, MARCUS HULTMARK, ALEXANDER SMITS, Princeton University — It has proven difficult in the past to test the true response of a hot-wire anemometry system to velocity fluctuations. In this set of experiments, exploiting the unique capabilities of the Princeton Superpipe, a hotwire probe is exposed to a turbulent flow with a known spectral composition, but an adjustable frequency content. By comparing the measured result from the anemometer to the well-defined input, a transfer function (or Bode plot) of the true response to velocity fluctuations is obtained. Results indicate that for most commonly used anemometry systems the response measured by this process is reasonably close to that predicted by the square-wave test. However, in situations where precise measurements of turbulence are desired, we question the suitability of the 3dB drop-off to accurately characterise frequency response. One of the tested anemometers exhibits a measured frequency response to velocity fluctuations that differs substantially to the response indicated from electronic testing. Predicted errors due to frequency response for various anemometry systems will be presented for turbulent boundary layer measurements.

3:18PM S7.00002 Turbulent Channel Flow Measurements Using Matched Hot-Wires

BARAHEH ESTEJAB, SEAN BAILEY, University of Kentucky — We present an experimental study conducted in a turbulent channel flow facility using hot-wire probes with both constant and varying viscous-scaled wire length. The objectives of the study were threefold: first, to validate the flow produced by the channel flow facility; second, to investigate the validity of recently proposed spatial filtering corrections for Reynolds stress profiles; and third, to extend the investigation of the near-wall peak Reynolds number dependence in turbulent pipe flow conducted by Hultmark, Bailey and Smits (see J. Fluid Mech. (2010), vol. 649, pp. 103–113). We found that in channel flow, unlike in the pipe flow experiments, the near-wall peak exhibited the same Reynolds number dependence observed in turbulent boundary layer studies and channel flow DNS. Since the same measurement techniques and procedures were used in the current study as used in the pipe flow study, this demonstrated that the near-wall Reynolds number independence observed in the pipe study was not due to error introduced by measurement methodology. Furthermore, comparison of results from wires of different length verified that spatial filtering corrections work in channel flow as well as pipe and boundary layer flows. Corrected results were in good agreement with channel flow DNS, thus verifying that the flow in the facility approximates one-dimensional turbulent Poiseuille flow.

3:31PM S7.00003 Uncertainties in interpretation of data from turbulent boundary layers due to measurement errors

RICARDO VINUESA, HASSAN NAGIB, IIT, Chicago — Composite expansions based on log law and power law were used to generate synthetic velocity profiles of ZPG turbulent boundary layers in the range $800 \leq Re_\theta \leq 8.6 \times 10^5$. Several artificial errors were then added to the velocity profiles to simulate dispersion in velocity measurements, error in determining probe position and uncertainty in measured skin friction. The effects of the simulated errors were studied by extracting log-law and power-law parameters from all these pseudo-experimental profiles, regardless of their original overlap region description. Various techniques were used, including the diagnostic functions (Ξ and Γ) and direct fits to logarithmic and power laws, to establish a measure of the deviations in the overlap region. The differences between extracted parameters and their expected values are compared for each case, with different magnitudes of error, to reveal when the pseudo-experimental profile leads to $ambiguous$ conclusions; i.e., when parameters extracted for log law and power law are associated with similar levels of deviations. This $ambiguity$ was observed up to $Re_\theta = 16,000$ for a 3% dispersion in the velocity measurements and $Re_\theta = 2,000$ when the skin friction was overestimated by only 2%. With respect to the error in the probe position, an uncertainty of 400 μm made even the highest Re profile ambiguous. The results from the present study are valid for air flow at atmospheric conditions.

3:44PM S7.00004 Reimagining the description of the turbulent boundary layer

RICHARD DUNCAN, HASSAN NAGIB, Illinois Institute of Technology — The turbulent boundary layer has been described from many different perspectives, drawing on physical inference, mathematical interpretation, and even visual observation. Many descriptions, however, lack complete explanation for the various features of the mean flow, or rely heavily on the description of only the streamwise mean velocity without analogous description of the Reynolds stress profiles. Here, a new description is proposed in which three regions of the turbulent boundary layer are identified, each defined by distinct physical phenomena and distinct mathematical solutions. A viscous region is shown to exist nearest the wall, with an extent further from the wall than previously thought. An inertial region, encompassing the potential region of the solution as well as the outermost part of the boundary layer, is proposed to help clearly and uniquely identify the extent of the boundary layer as well as allow for description of other wall-bounded flows, which possess a different outer boundary condition. Finally, the region in which turbulence is dominant is singled out as a separate region, calling into question the classical inner versus outer scaling while better explaining the presence of intermediate regions such as the classical buffer layer and wake region.

3:57PM S7.00005 Self-similar vorticity apportionment in turbulent wall-flows , JOE KLEWICKI, University of New Hampshire and The University of Melbourne — The analytical closure by Fife et al. (*J. Disc. & Cont. Dyn. Sys.* **24**, 2009) allows the mean momentum equation for turbulent wall-flows to be represented by an invariant set of nonlinear ordinary differential equations. With appropriate starting conditions, these equations are integrated over an internal domain specified by the theory, and yield solutions for the mean velocity, Reynolds stress and their derivatives. The present talk primarily investigates the affiliated similarity structure of the mean vorticity field, and, in particular, its development as a function of Reynolds number. Existing data from boundary layers ($300 < \delta^+ < 50,000$), pipes ($180 < \delta^+ < 530,000$) and channels ($180 < \delta^+ < 5,000$) are shown to exhibit the theoretically predicted mean vorticity decay rate scalings. The outward movement of the centroid of the mean vorticity distribution (with δ^+) into a region dominated by turbulent inertia is shown to coincide with the onset of the asymptotic (four- layer) dynamical regime. Evidence supporting the emergence of a self-similar relationship between the mean and rms spanwise vorticity is clarified through their relationships to the length scale distribution intrinsic to the mechanism of turbulent inertia. Overall, the results of the theory and data are discussed relative to two physically distinct mechanisms by which the velocity and vorticity field motions exhibit scale separation with increasing Reynolds number.

4:10PM S7.00006 A New Scaling for Adverse Pressure Gradient Turbulent Boundary Layers[1] , FLINT THOMAS, University of Notre Dame, DAVID SCHATZMAN, NASA Ames Research Center — A new scaling for strong adverse pressure gradient (APG) turbulent boundary layers (TBL) is presented. The new scaling is applied to data from the author's APG TBL experiment as well as several previous experimental studies. Both steady and unsteady flows are considered. The new scaling is shown to provide an excellent collapse of not only the mean velocity but also the turbulent stress profiles. The physical motivation for the scaling is presented in terms of underlying stability mechanisms as evidenced by a series of conditional boundary layer measurements. The implications of the scaling on the physics of strong APG TBL flows is also discussed.

[1]This work was supported by the U.S. Army Research Office under W911NF-07-1-0122.

4:23PM S7.00007 Non-universal k_1^{-1} laws in pressure-gradient-driven turbulent wall-bounded flows , O. RAMESH, Indian Institute of Science, SHIVSAI DIXIT, MKSSS's Cummins College of Engineering for Women, Pune, India — The scaling laws for the spectra of streamwise velocity fluctuations in turbulent wall-bounded flows are reformulated with inclusion of the streamwise pressure gradient. These laws indicate that the presence of pressure gradient naturally leads to the so-called incomplete similarity of spectra irrespective of the mean flow acceleration. Interestingly however, the corresponding spectral overlap arguments still lead to the inverse-power variation of the power spectral density of streamwise velocity fluctuations i.e. the k_1^{-1} law. These are, however, the non-universal k_1^{-1} laws arising out of incomplete similarity. Experimental evidence in the literature on pipe and channel flows clearly supports this. Striking experimental evidence is presented in favour of the non-universal k_1^{-1} laws for an accelerating turbulent boundary layer flow. It is observed that the prerequisite condition of "high Reynolds number" for having substantial spectral overlap in experiments appears to be remarkably relaxed in the presence of streamwise mean flow acceleration.

4:36PM S7.00008 Turbulence in Accelerating Boundary Layers[1] , PRANAV JOSHI, XIAOFENG LIU, JOSEPH KATZ, The Johns Hopkins University — This study focuses on favorable pressure gradient (FPG) turbulent boundary layers. 2D PIV data has been obtained in a sink flow in multiple planes and locations, at two Reynolds numbers and acceleration parameters. FPG decreases the locally normalized stresses over the entire boundary layer, but increases their magnitude close to the wall (y/δ<0.1) and decreases it in outer regions. This turbulence suppression is associated with confinement of coherent structures originated in the inner part of the boundary layer to a narrow near-wall region. Several contributors have been identified: (1) Weaker normalized strength of structures and consequently weaker self-induced wall-normal transport, (2) the negative ∂V/∂y and high ∂U/∂y, which preferentially orient inclined structures parallel to the wall. In the FPG region, almost all of the small-scale turbulence is confined to low speed streaks, where ejections bring fresh turbulence away from the wall, while sweeps bring in outer layer fluid with low turbulence. Pressure calculations based on integration of material acceleration obtained from time-resolved data show that ejections are associated with negative pressure gradients, as the upward moving fluid accelerates, while the opposite holds for sweeps.

[1]Sponsored by NSF.

4:49PM S7.00009 Direct numerical simulation of turbulent boundary layers under unsteady pressure gradients[1] , WILLIAM BROMBY, DONGHYUN YOU, Carnegie Mellon University — Direct numerical simulations are performed to improve the understanding of unsteady separation processes of turbulent boundary layers characterizing the performance and efficiency of many aerodynamic applications such as helicopter rotor blades, wind turbine blades, pitching and flapping airfoils and wings, and rotating turbomachines. A time varying blowing-suction velocity distribution is imposed along the upper boundary to introduce unsteady adverse pressure gradients to the turbulent boundary layer. The distinct characteristics of turbulent boundary layers under unsteady adverse pressure gradients including unsteady boundary-layer detachment and reattachment, and production and dissipation of turbulent kinetic energy and vorticity, are revealed by a systematic comparison with steady attached/separated turbulent boundary layers.

[1]Supported by the Army Research Office Grant W911NF1010348

5:02PM S7.00010 Turbulent Boundary Layers Absent Mean Shear , BLAIR JOHNSON, EDWIN COWEN, Cornell University — In environmental flows, we often observe turbulence levels that far exceed those produced by mean boundary shear (e.g., breaking surface waves), contributing to significant sediment resuspension and turbulent boundary layers that differ strikingly from classic turbulent boundary layer characterizations. We choose to study sediment resuspension in turbulence in the absence of mean shear using a novel laboratory facility that uses a Randomly Actuated Synthetic Jet Array (RASJA), designed to generate homogeneous isotropic turbulence with low mean flows. We use Particle Image Velocimetry (PIV) at both solid glass and sediment boundaries to examine the turbulent structures and nature of the flow at the two bed conditions. The sediment boundary is narrowly graded sand with a median grain size (D50) of 260 microns. Surprisingly, we find that the interaction of turbulence with a sediment boundary results in the formation of ripple patterns, with spacings on the order of the integral length scale. We measure metrics such as the turbulence intensity, turbulent kinetic energy, spectra, and dissipation. Our analysis includes a quadrant-based Reynolds stress analysis and the determination of critical turbulent stresses responsible for sediment resuspension, from which we develop a non-dimensional Shields-like parameter that captures incipient particle motion.

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S12 Turbulence Simulation VI 315

3:05PM S12.00001 Error Quantification in Simulations of Variable Density Low Mach Number Turbulent Flows , NICHOLAS ROBERTSON, SIDDHARTHA VERMA, GUILLAUME BLANQUART, California Institute of Technology — In numerical simulations of low Mach number flows, the density is often expressed as a function of one or several transported scalar quantities through the use of an equation of state (EOS). As a result, these scalars play a critical role in controlling the accuracy of the overall simulation. This study aims at understanding and quantifying the sources of errors introduced in performing simulations of variable density low Mach number laminar and turbulent flows. As a first step, the relative importance of the order of accuracy of the scalar transport scheme and the velocity scheme is analyzed in canonical variable density test cases. Then, various implementations of the EOS are investigated in laminar flows. Particular importance is placed on the robustness, the convergence, and the accuracy of these implementations. Finally, the different scalar transport schemes and implementations of the EOS are combined and evaluated in simulations of turbulent planar jets. Following this analysis, guidelines for performing accurate and robust simulations of variable density low Mach number laminar/turbulent flows are proposed.

3:18PM S12.00002 Large-Eddy Simulation of Crashback in a Ducted Propulsor , HYUNCHUL JANG, KRISHNAN MAHESH, University of Minnesota — Crashback is an operating condition to quickly stop a propelled vehicle, where the propeller is rotated in the reverse direction to yield negative thrust. The crashback condition is dominated by the interaction of free stream flow with strong reverse flow. Crashback causes highly unsteady loads and flow separation on blade surface. This study uses Large-Eddy Simulation to predict the highly unsteady flow field in crashback for a ducted propulsor. Thrust mostly arises from the blade surface, but most of side-force is generated from the duct surface. Both mean and RMS of pressure are much higher on inner surface of duct, especially near blade tips. This implies that side-force on the ducted propulsor is caused by the blade-duct interaction. Strong tip leakage flow is observed behind the suction side at the tip gap. The physical source of the tip leakage flow is seen to be the large pressure difference between pressure and suction sides. The conditional average during high amplitude event shows that the tip leakage flow and pressure difference are significantly higher. This work is supported by the United States Office of Naval Research under ONR Grant N00014-05-1-0003.

3:31PM S12.00003 Investigating the effect of hull on marine propeller in crashback using LES , AMAN VERMA, HYUNCHUL JANG, KRISHNAN MAHESH, University of Minnesota — Crashback is an off-design condition where the marine vessel moves forward while the propeller is in reverse rotation, resulting in a highly unsteady separated flow. According to experiments, an upstream hull significantly increases the side-forces on a propeller in crashback below an advance ratio of $J = -0.7$. LES is performed for an open propulsor with and without hull at two advance ratios, $J = -1.0$ and $J = -0.5$. The experimentally observed behavior is reproduced using LES. At $J = -1.0$, two noticeable flow features are found with the hull - a recirculation zone located on the hull in the vicinity of the propeller and a vortex ring much closer to the propeller. In contrast, at $J = -0.5$, the recirculation zone is weaker and farther upstream of the propeller. As a result, the hull does not significantly alter the flow in the immediate vicinity of the propeller. For both advance ratios, side forces are mostly generated from the leading edge separation on suction side. However, high levels of side forces are also generated from trailing edge separation on suction side at $J = -1.0$. This work was supported by the United States Office of Naval Research under ONR Grant N00014-08-1-0433.

3:44PM S12.00004 Marine Propeller Analysis with an Immersed Boundary LES methodology[1] , SETH SCHROEDER, Naval Surface Warfare Center Carderock Division, ELIAS BALARAS, The George Washington University — Modern marine propeller design and analysis techniques employ a wide range of computational methods to balance time constraints and accuracy requirements. Potential flow and RANS based methods have historically comprised the tools necessary for design and global performance analysis. However, for analysis where unsteady flow phenomena are of interest, eddy resolving methodologies are required. In the present study we use the large-eddy simulation (LES) approach coupled with an immersed-boundary (IB) method to perform computations of the flow around a rotating propeller at laboratory Reynolds numbers. Compared to classical boundary conforming strategies our formulation eliminates meshing overhead time and allows for body motion without additional treatments such as overset or dynamic meshing. The structured Cartesian grid also allows for a non-dissipative solver which conserves mass, momentum and energy. We will focus on the Italian Ship Model Basin (INSEAN) E1619 propeller and compare the predictions of our method to experimental results. The E1619 propeller is a 7-bladed submarine stock propeller that has been the subject of extensive experimental testing and previous computational studies.

[1]Supported by ONR

3:57PM S12.00005 Turbulence Simulation using many Graphics Processors[1] , ALI KHAJEH-SAEED, J. BLAIR PEROT, University of Massachusetts Amherst, Mechanical and Industrial Engineering, Amherst, MA 01003, United States — Unsteady simulations of turbulence are performed using up to 64 graphics processors on the NSF XSede supercomputer, Lincoln, located at NCSA. For a 512^3 simulations the performance of 16 GPUs (Tesla S1070) is about 45 times faster than that obtained with the same number of CPU cores of quad-core Intel Harpertown processors on the same machine. The code is optimized to use the fast shared-memory on the GPUs and to use communication/computation overlapping. Results show that the computation time is now so fast that even for large problems, with up to 8 million unknowns per GPU, the MPI communication time controls the scaling behavior of the CFD algorithm.

[1]This work is supported by the Department of Defense and Oak Ridge National Laboratory.

4:10PM S12.00006 Development and testing of a user-friendly Matlab interface for the JHU turbulence database system[1] , JASON GRAHAM, Mechanical Engineering and CEAFM, The Johns Hopkins University, EDO FREDERIX, Permanent address: Mechanical Engineering, Eindhoven University of Technology, The Netherlands, CHARLES MENEVEAU, JHU — One of the challenges that faces researchers today is the ability to store large scale data sets in a way that promotes easy access to the data and sharing among the research community. A public turbulence database cluster has been constructed in which 27 terabytes of a direct numerical simulation of isotropic turbulence is stored (Li et al., 2008, JoT). The public database provides researchers the ability to retrieve subsets of the spatiotemporal data remotely from a client machine anywhere over the internet. In addition to C and Fortran client interfaces, we now present a new Matlab interface based on Matlab's intrinsic SOAP functions. The Matlab interface provides the benefit of a high-level programming language with a plethora of intrinsic functions and toolboxes. In this talk, we will discuss several aspects of the Matlab interface including its development, optimization, usage, and application to the isotropic turbulence data. We will demonstrate several examples (visualizations, statistical analysis, etc) which illustrate the tool.

[1]Supported by NSF (CDI-II, CMMI-0941530) and Eindhoven University of Technology's Masters internship program.

4:23PM S12.00007 Large eddy simulation of decaying homogeneous isotropic turbulence on a block-structured non-uniform grid using reconstruction , LAUREN GOODFRIEND, TINA CHOW, UC Berkeley, MARCOS VANELLA, ELIAS BALARAS, George Washington University — Modeling turbulence directly requires too many computational resources for many realistic problems to be tractable, so methods have been developed to reduce the computational size of turbulence simulation. Large-eddy simulation and adaptive mesh refinement reduce the computational cost of turbulence modeling compared to direct numerical simulation, but they are rarely used together. Errors appear at grid refinement interfaces due to dependence of computed quantities on the filter width and insufficient smoothness of the solution at the grid scale. Here, approximate reconstruction of the unfiltered velocity field is used to decrease these errors in a simulation of decaying homogeneous isotropic turbulence advected past a grid refinement interface. Different explicit filter types and levels of reconstruction are tested. Reconstruction was found to improve results in cases using a binomial approximation to a Gaussian filter for a fine to coarse grid refinement interface, the natural transition for decaying turbulence. These results inform the use of large-eddy simulation on more complicated adaptive mesh refinement grids.

4:36PM S12.00008 One-way coupling of nested grids in WRF-LES and local LES for turbulent boundary layer , TETSURO TAMURA, Tokyo Institute of Technology, KOJIRO NOZAWA, Shimizu Corporation — This study proposes the method of one-way coupling between nested grids with different resolution for LES of turbulent boundary layer. Based on the difference of the momentum equations for the coarse and fine grids, we deduce the equation of the finer-grid component of velocity fluctuations. This equation has two types of SGS stress terms. One is the energy cascade from finer-grid GS turbulence to SGS turbulence and the other is the production term in this equation which cascades turbulence energy from coarser grid to finer grid. In order to validate the proposed method, we consider the a-priori test. Coarser-grid velocity is obtained by filtering the solution computed on finer grid. There are two types of finer-grid component of velocity, such as the predicted values by solving the proposed equation or the exact values. Comparison has been performed for the results of rough-wall turbulent boundary layer simulation. The spectra of velocity fluctuation obtained as sum of the predicted finer-grid component and the coarser-grid velocity appropriately fit to the -5/3 power law for the inertial sub-range. As posteriori test, the proposed method is also applied to atmospheric flows nesting WRF-LES and local LES. Coarser-grid velocity is obtained by WRF-LES. Finer-grid solutions are checked and the potential of the proposed method is investigated.

4:49PM S12.00009 Designing LES of the High Reynolds Surface Layer to Account for Numerical Friction in the Algorithm. , JAMES BRASSEUR, GANESH VIJAYAKUMAR, Penn State Univ, MATTHEW CHURCHFIELD, NREL, ADAM LAVELY, Penn State Univ, ERIC PATERSON, PATRICK MORIARTY, NREL — Numerical friction stabilizes large-eddy simulation (LES), but also impacts accuracy. We explore this issue using a theory (Brasseur & Wei 2010) where the LES is designed in a 3-parameter space that quantifies the level of friction in the SFS stress model (Re$_{LES}$), the relative content of resolved to SFS stress ($\Re$), and surface layer resolution. To achieve law-of-the-wall in the mean, the LES must be in the "high-accuracy zone" (HAZ) of the $\Re - Re_{LES}$ parameter space. Using rough-wall channel flow and atmospheric boundary layer LES, we analyze simulations that are identical except for spectral vs. finite volume (FV) algorithms. Numerical friction shifts the LES away from the HAZ in the $\Re - Re_{LES}$ parameter space consistent with changes in mean shear-rate. The effective low pass filter from numerical friction shifts the total stress from resolved to subfilter-scale contributions, and effect that is more apparent when the spectral version of the LES is in the HAZ. A consequence is the enhancement of streamwise coherence in turbulence structure, particularly apparent in the integral scales. We shall discuss the requirements to adjust the FV LES to match a corresponding spectral LES in the HAZ, and differences in efficiency and accuracy. *Support: NSF, DOE*.

5:02PM S12.00010 Local relaxation and maximum entropy states in two-dimensional decaying turbulence , SERGIO SERVIDIO, Universita della Calabria, MINPING WAN, WILLIAM H. MATTHAEUS, University of Delaware, VICENZO CARBONE, Universita della Calabria — The phenomenon of vortex merging in two-dimensional turbulence has been investigated through direct numerical simulations. The fast and local processes that occur during the turbulent relaxation of a randomly initialized system in periodic geometry have been examined. The analysis reveals that many of the coherent structures can be described by a local principle of maximization of entropy. The validity of this entropy principle has been further confirmed by time-dependent statistics using a contour-tracking technique. Implications for the description of persistent coherent vortices commonly observed in nature are suggested, including growing evidence for the wide applicability of maximum entropy-based relaxation principles. [Phys. Fluids 22, 125107 (2010)].

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S16 Porous Media IV 319

3:05PM S16.00001 Mass Transport Measurement in 3D Porous Media for Carbon Sequestration , ROBERT ECKE, SCOTT BACKHAUS, Los Alamos National Laboratory, KONSTANTIN TURITSYN, MIT — To determine the storage potential for carbon sequestration strategies involving porous media, accurate determination of mass transport efficiency is required. We have made accurate measurements of mass transport in geometries similar to those relevant for sequestration, namely a gravitationally stable two layer system where the diffusion interface between the two phases is unstable. We use water and propylene glycol as the fluids in cells with cylindrical geometry. For a range of Rayleigh numbers between 150 and 5000, we find mass transport efficiency that is consistent with steady state thermal heat transport convection in porous media and with 2D mass transport in a Hele-Shaw geometry [1]. A transition from a high mass transport state to a lower mass transport state typically occurs between 4 and 6 convective times and is observed over the full Ra range.

[1] S. Backhaus, K. Turitsyn, and R.E. Ecke, Physical Review Letters **106**, 104501 (2011).

3:18PM S16.00002 Evaporation from a semi-infinite porous medium: The role of capillary flow , H.K. NAVAZ, B. MARKICEVIC, Kettering University, S.J. PAIKOFF, DTRA — The liquid evaporation from the semi-infinite porous medium is solved numerically using the dynamic capillary network model in which the interface shape and multiphase flow front thickness between dry and fully wet parts of porous medium are tracked in time. Both convective and diffusion mass transport limited regimes are identified and liquid pseudo-velocity due to the evaporation is calculated. The numerical analysis is extended for in-parallel capillary flow and evaporation liquid transport, and again, the changes of the interface shape and multiphase flow front thickness are investigated. It turns out that the convective evaporation is prolonged due to the capillary flow as evaporated liquid close to the evaporating boundary is replenished by capillary flow. However, the evaporation curve has an elongated "tail" for longer evaporation times as capillarity tends to transport the liquid deeper into the porous medium. The contributions of the capillary flow and the mass transport on the overall evaporation dynamics is best visible by comparing the liquid pseudo-velocity for pure evaporation and evaporation with capillary flow. Two pseudo-velocities are equal for time for which there is a transition from convection to diffusion controlled evaporation. In this point, the remaining liquid is always distributed in the multiphase pattern, where the thickness of the multiphase region depends on capillary flow and mass transport rates.

3:31PM S16.00003 New upper bounds for convection in a fluid-saturated porous layer[1] , BAOLE WEN, GREG CHINI, University of New Hampshire, CHARLES DOERING, University of Michigan — There has been renewed interest in buoyancy-driven convection in porous media owing in part to applications relating to carbon dioxide sequestration in terrestrial aquifers. As in other convection problems, a key quantity of interest is the normalized volume and time averaged heat flux through the layer, i.e. the Nusselt number (Nu). Here, we present an improved upper bound on Nu as a function of Rayleigh number (Ra) for a model of thermally driven porous medium convection. The bound is obtained by numerically solving the full "background field" variational problem first posed by Doering and Constantin (1998) for this model. We describe an efficient numerical algorithm for solving the variational problem and present improved bounds indicating that Nu $\sim$ c Ra (for computed constant c).

[1]GC gratefully acknowledges support for this work through NSF grant 0928098.

3:44PM S16.00004 Gravity Current in Horizontal Porous Media with A Permeability Gradient[1] , ZHONG ZHENG, PEICHUN TSAI, TALAL AL-HOUSSEINY, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University, COMPLEX FLUID GROUP TEAM — We study the influence of a power-law porosity and permeability gradient on the front propagation of a gravity current in an unconfined porous media. We neglect mass transfer and surface tension on the interface. A similarity solution is found for the propagating front, which is different from the homogeneous case. Experiments have been performed using liquid pushing air in a Hele-Shaw cell with a constant gradient in gap thickness in the vertical direction. We measure the speed of the front and the shape of the interface. We observe a third layer of trapped air in the region where the permeability is low, while it appears that the propagating front still satisfies the similarity solution with a modified coefficient.

[1]This work is supported by a funding from Carbon Mitigation Initiative at Princeton University

3:57PM S16.00005 PIV measurement of solute diffusion natural convection in a Hele-Shaw cell , DANA EHYAEI, MATTHEW BALDUS, KEN KIGER, University of Maryland — Dissolution trapping of CO_2 is regarded as a critical mechanism for sequestration of carbon in deep saline aquifers. Currently, however, there is significant variability in the literature concerning the critical time required for onset of the convective fingering instabilities that are responsible for accelerating the dissolution of CO_2 into the brine. In the current work, we focus on making a direct measure of the velocity field during the onset, growth and saturation of the convection in a Hele-Shaw cell analog of the porous media problem. The working fluids consisted of aqueous mixtures of methanol and ethylene glycol (MEG), which when dissolved into water, results in a negatively buoyant solution similar to CO_2 in brine. The velocity field was measured using PIV, and procedures were developed to circumvent traditional implementation difficulties within thin-gap flows through careful particle selection and manipulation prior to starting the experiment. The results of these measurements provide the unsteady evolution of the flow from onset to late-time mixing, as revealed by the velocity and vorticity field during the finger growth and saturation. Features of the flow and the rate of mixing will be discussed.

4:10PM S16.00006 Transient double-diffusive convection in an axisymmetric wavy-walled enclosure[1] , FAUSTO SANCHEZ, SIMON MARTINEZ, HUGO RAMIREZ, JESUS CHAVEZ, Universidad Autonoma de Nuevo Leon, ABRAHAM MEDINA, Instituto Politecnico Nacional — Transient double-diffusive convection within a vertical wavy-wall cylinder was numerically studied. The enclosure is filled with a saturated porous medium, the fluid is Newtonian and the Boussinesq approach was applied. The cavity is heated from below while the lateral wavy wall is thermally insulated. The saturation concentration of a passive tracer is assumed to exist at the top. The initial condition considers the fluid is at rest with zero concentration. An analytical coordinate transformation was used to change the computational domain into a square. Heat and mass transfer were analyzed using non-dimensional parameters which include the cavity aspect ratio, dimensionless wavelength and amplitude of the wavy-wall, the Rayleigh-Darcy, effective Prandtl and Schmidt numbers. The average Nusselt and Sherwood numbers were evaluated while the transport phenomena evolve within the cavity. After a large dimensionless time the concentration gradients become weak and thermal stratification arises promoting low heat transfer, specially when the wave amplitude is large.

[1]Authors thank CONACyT Projects 62054 and 103334, and PAICyT-UANL.

4:23PM S16.00007 Gravity currents arrested by convective mixing , CHRISTOPHER MACMINN, MICHAEL SZULCZEWSKI, JUAN HIDALGO, RUBEN JUANES, Massachusetts Institute of Technology — The large-scale injection of carbon dioxide (CO2) into deep saline aquifers is a promising tool for reducing atmospheric CO2 emissions to mitigate climate change. Trapping of the buoyant CO2 after injection is essential in order to minimize the risk of leakage into shallower formations through a pre-existing well or fracture, or via the activation of a fault. However, traditional reservoir-simulation tools are currently unable to resolve the impact of small-scale trapping processes on fluid flow at the scale of a geologic basin. Here, we use analog experiments to study solubility trapping of the CO2 via convective mixing, where dense fingers of CO2-rich groundwater carry CO2 away from the buoyant plume as it dissolves. We study the impact of convective mixing on a buoyant gravity current migrating up-dip in a sloping aquifer (a Hele-Shaw cell packed with glass beads), and we show that a simple upscaled model reproduces the macroscopic features of this complex physical process both qualitatively and quantitatively. We then estimate the dimensionless rate of solubility trapping for several large saline aquifers in the United States in order to assess the importance of solubility trapping in practice.

4:36PM S16.00008 Density-Driven Convection with an Inclined Boundary , PEICHUN TSAI, TALAL AL-HOUSSEINY, ZHONG ZHENG, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University — We experimentally investigate convective instability and transport in a Hele-Shaw geometry and in a porous medium with inclined boundaries. The initial fluid configuration is buoyantly stable: a lighter liquid is placed above a dense one. The convection is generated by the dissolution of the two liquids which results in a heavier fluid layer at the interface, advecting into the underlaid, lighter fluid. Phenomenologically, in a vertical cell, heavier, mushroom-like plumes mostly travel downward and then partially merge. In contrast, in an inclined cell, the dense fingerings initially transport vertically downward and subsequently tilt and move laterally due to the inclined boundaries. We examine the width, wavelength, and dynamics of the dense plumes. We find that the tilting angle of the inclined boundary profoundly affects the dynamics of the density-driven plumes. In addition, the permeability of the porous media strongly changes the convective rate. These findings show key implications for geological carbon dioxide (CO2) storage in a silane aquifer when the dissolved CO2 into brine produces a heavier mixture with an enhancement of the mass transfer by convection.

4:49PM S16.00009 How convective mixing slows down , MICHAEL SZULCZEWSKI, RUBEN JUANES, Massachusetts Institute of Technology — Convective mixing is a key CO2-trapping mechanism during geologic sequestration. While this mechanism has been shown to increase the CO2 dissolution rate at short times after onset, it has not been studied at late times when the CO2-rich fingers interact with the bottom of the aquifer. Here, we study the late-time behavior in a simple system: a linear, homogeneous aquifer in which the CO2-brine interface spans a finite region along the top of the aquifer. We perform high-resolution simulations and experiments that involve dissolving CO2 in a Hele-Shaw cell filled with water and a pH indicator. We show that the late-time dissolution rate decreases after the fingers reach the bottom due to two mechanisms: (1) fill up, in which the CO2-rich fingers accumulate beneath the free-phase CO2, decreasing the density difference that drives convective mixing; and (2) shielding, in which the accumulation of CO2-rich fluid forms a wedge that blocks water without CO2 from reaching the entire CO2-brine interface. We further show that the dissolution rate decreases according to a power law in time. These results will be helpful for calculating the timescale over which a volume of injected CO2 will completely dissolve.

5:02PM S16.00010 Purple fingers: convection in carbon sequestration , ANJA SLIM, MAHESH BANDI, L. MAHADEVAN, Harvard University — In geological carbon sequestration, CO_2 injected into a saline formation is less dense than the resident brine and floats above it. It is slightly soluble in brine and progressively dissolves. Brine with dissolved CO_2 is slightly denser than "pure" brine giving the potential for convective overturning. We experimentally study the dissolution process from first contact between CO_2 and brine through onset of convection to almost complete saturation for Rayleigh numbers between 80 and 1200. We present an analogue experiment using potassium permanganate as our solute, which closely mimics relevant CO_2 properties. We describe the different dynamical regimes and connect these with trends in several global measures, including the dissolution flux. We find onset of convection using both amplitude and flux deviations from pure diffusion.

Tuesday, November 22, 2011 3:05PM - 5:02PM –
Session S17 Reacting Flows V: Laminar Flames & Instabilities 320

3:05PM S17.00001 A computational method for the solution of the edge flame velocity eigenvalue as a two-dimensional boundary value problem with applications to realistic heat release , KAI-PIN LIAO, MOSHE MATALON, CARLOS PANTANO, University of Illinois at Urbana-Champaign — We present a new numerical method to determine the edge flame velocity in a counterflow as an eigenvalue of the two-dimensional boundary-value problem for the variable density equations in the zero Mach number limit. The method utilizes a collocated arrangement of all variables in space and relies on discrete mass conservation using centered second-order accurate finite-differences. The finite element method approach, weak form, is adopted to determine the discretization near boundary and ensure well-posedness of the equations. Pressure and velocities are coupled and solved iteratively, while energy and species equations are segregated and solved sequentially. The method is coupled with pseudo-arc length continuation to explore the full parametric dependence of the solution. The edge-flame velocity and structure under the combined effect of strain and heat release will be presented.

3:18PM S17.00002 Slow invariant manifolds for reaction-diffusion systems , JOSHUA MENGERS, JOSEPH POWERS, University of Notre Dame — Slow Invariant Manifolds (SIMs) are calculated as a reduction technique for isothermal closed reaction-diffusion systems. Reaction-diffusion systems are stiff infinite dimensional dynamical systems that are costly to simulate. These systems' dynamics are reduced by using a Galerkin projection onto a finite-dimensional approximate inertial manifold, and then further reduced by projection onto a low-dimensional SIM. The Galerkin projection rigorously accounts for the coupling of the reaction and diffusion processes. The SIM accurately isolates the system's long time dynamics. A robust method of constructing one-dimensional SIMs by calculating equilibria and then integrating to find heteroclinic orbits is extended to systems with weak spatial heterogeneity. Examples are demonstrated on simple physical reaction mechanisms.

3:31PM S17.00003 Transient OH* Chemiluminescence Imaging of Acoustically Coupled Fuel Droplet Combustion[1] , JEFFREY WEGENER, CRISTHIAN SEVILLA, JENNIFER SMOLKE, AARON SUNG, KELVIN CHEN, OWEN SMITH, ANN KARAGOZIAN, UCLA — This study focuses on combustion of liquid fuel droplets during exposure to external acoustic disturbances generated as standing waves within a closed acoustic waveguide. During such acoustic excitation, the mean flame orientation is observed to be dependent on the droplet's location relative to the pressure node (PN), and is consistent with the sign of a theoretical acoustic acceleration acting on the burning system. Yet experimentally estimated acoustic accelerations, measured from the degree of mean flame deflection, differ substantially in a quantitative sense from those predicted by theory.[2] Phase-locked OH* chemiluminescence imaging reveals a deflected flame which oscillates in position relative to the droplet, with the largest degree of oscillation near the PN. A range of acoustic forcing frequencies and droplet locations are used to investigate flame movement over multiple acoustic cycles. The degree of flame oscillation, mean flame deflection angle, and fuel droplet burning rate all correlate with one another for different relative positions of the droplet.

[1]Supported by AFOSR.
[2]Tanabe, et al., **Proc. Comb. Inst.**, 2000

3:44PM S17.00004 Puffing flame instability - Part I: Numerical investigation and analysis[1] , SHYAM MENON, GUILLAUME BLANQUART, Mechanical Engineering, California Institute of Technology, PHILIPP BOETTCHER, JOSEPH SHEPHERD, Graduate Aerospace Laboratories, California Institute of Technology — Experimental and computational work has been carried out to understand hot-surface ignition and the subsequent flame propagation in low pressure environments. In the experiments, a "puffing" instability is observed at rich hydrocarbon fuel-air mixture ratios. The generation and transport of these flame "puffs," characterized by vortex rings, is investigated in this work by combining results from numerical simulations and experiments. Analysis of the sources of vorticity shows that baroclinic torque is the major contributor to vorticity generation. Misalignment of buoyancy induced pressure gradients in the flow field with density gradients at the flame front leads to the formation of vortex rings. Circulation of these individual vortex rings is calculated and correlated to the formation time scale of a flame "puff." The effect of different parameters such as gravity, flame speed, and density ratio on the "puffing" frequency is investigated and a correlation of the puffing frequency with these parameters is proposed.

[1]Research supported by the Boeing Company.

3:57PM S17.00005 Puffing flame instability - Part II: Predicting the onset and frequency[1] , PHILIPP BOETTCHER, JOSEPH SHEPHERD, SHYAM MENON, GUILLAUME BLANQUART, California Institute of Technology — Experiments and simulations have been performed on fuel rich n- hexane air mixtures in a closed vessel. Both experiments and simulations show a distinct cyclic combustion or "puffing" mode. The misalignment of buoyancy induced pressure gradients and density gradients across the flame front is responsible for the generation of vorticity and its subsequent roll-up into vortex rings. In the present work, a simplified model is proposed based on the fundamental interactions between fluid mechanical and chemical parameters. This simplified fluid mechanics model is based on dimensional analysis and is used to predict the onset and frequency of the puffing behavior.

[1]This work was sponsored by The Boeing Company through CTBA-GTA-1.

4:10PM S17.00006 Comparing a Rayleigh-Taylor Unstable Flame to a Circular Cylinder , ELIZABETH HICKS, University of Chicago/ Northwestern University — Rayleigh-Taylor unstable, premixed flames show different types of behavior depending on the value of the gravitational force. We present the results of two-dimensional numerical simulations of the flame for various values of gravity. If gravity is relatively small, the flame forms a cusped shape. The cusped flame produces stable shear layers, or rolls, behind the flame. For slightly higher values of gravity, the rolls become unstable and vortex shedding begins far behind the flame front. We show that the development of this vortex shedding behavior can be described by the Landau equation and can be represented dynamically by a Hopf bifurcation. This behavior is analogous to the initial instability behind a circular cylinder, which leads to the von Karman vortex street for large enough values of the Reynolds number. The applicability of the Landau equation allows the apparently complex spatio-temporal characteristics of the vortex shedding to be simply described temporally with only a secondary spatial dependence.

4:23PM S17.00007 Magnetic field effects on convective instability of autocatalytic fronts , MANORANJAN MISHRA, Indian Institute of Technology Ropar, 140001 Ropar, India, ANDRE THESS, Ilmenau University of Technology, 98684 Ilmenau, Germany, ANNE DE WIT, Université Libre de Bruxelles, 1050 Brussels, Belgium — The coupling between autocatalytic chemical reactions and diffusion can lead to traveling fronts whereby products invade fresh reactants at a constant speed. The properties of these reaction-diffusion fronts have been studied in details experimentally in gels used to avoid any convective motions. In absence of gels, the dynamics can be perturbed by buoyancy-driven convective flows related to density changes across the front. Experiments have shown that magnetic fields can modify such convective dynamics of traveling fronts. In this context, we study theoretically the influence of a magnetic field on the density fingering pattern of such a reaction-diffusion-convection system. The magnetization is oriented perpendicularly to the plane in which the front travels. The solutions of both products and reactants are assumed diamagnetic (i.e. negative magnetic susceptibility). The influence of the magnetic field is analyzed by numerical simulations of the reaction-diffusion-convection equations for the concentration of the autocatalytic product coupled to Darcy's law for the flow velocity through a concentration dependent magnetic force. We show that the magnetic field is able to suppress or enhance the convective instability depending on the value of the magnetic Rayleigh number of the problem.

4:36PM S17.00008 ABSTRACT WITHDRAWN —

4:49PM S17.00009 ABSTRACT WITHDRAWN —

Tuesday, November 22, 2011 3:05PM - 5:02PM –
Session S18 Particle-laden Flows V 321

3:05PM S18.00001 Lagrangian statistics of light particles in turbulence , VIVEK N. PRAKASH, JULIAN MARTINEZ MERCADO, YOSHIYUKI TAGAWA, CHAO SUN, DETLEF LOHSE, Physics of Fluids Group, University of Twente — We study the Lagrangian velocity and acceleration statistics of light particles (bubbles in water) in homogeneous and isotropic turbulence. Bubbles of size comparable to the Kolmogorov length scale (Stokes numbers $\approx$ O(0.01)) are dispersed in a turbulent water tunnel operated at Taylor Reynolds numbers Re_λ ranging from 130 to 260. The bubble trajectories are experimentally captured using the three-dimensional Lagrangian Particle Tracking technique. The bubble acceleration PDFs are found to be highly intermittent with flatness values around 23-30. We study the dependence of the velocity and acceleration statistics (PDFs and autocorrelation functions) on Re_λ and compare our results to those from point-particle DNS and other experiments. We also present our latest results on the Lagrangian statistics of finite-size inertial bubbles (size $\approx$ 2-4 mm and St $\approx$ O(1)).

3:18PM S18.00002 Interpolation error in DNS simulations of turbulence: consequences for particle tracking , MICHEL VAN HINSBERG, JAN TEN THIJE BOONKKAMP, FEDERICO TOSCHI, HERMAN CLERCX, Eindhoven University of Technology, The Netherlands — An important aspect in numerical simulations of particle-laden turbulent flows is the interpolation of the flow field. Interpolations are needed in many applications, for example, simulations of turbulent aerosol transport, transport of bed load sediments and marine species. For the interpolation different approaches are used. Where some studies use low-order linear interpolation, others use high-order spline methods. This study focuses on estimating the interpolation error and compares it with the discretisation error of the flow field. In this way one can balance the errors in order to achieve an optimal result. Algorithms have been developed for the approximation of the interpolation error. As a spin-off of the theoretical analysis a practical method is proposed which enables direct estimation of the interpolation error from the energy spectrum of the flow. Furthermore it is shown how this energy spectrum is affected by the interpolation. Our results suggest that a coherent choice of the interpolation method and the value of $k_{max}\eta$ should be made in order to balance the errors. Our approach may provide a quantitative indicator for this purpose.

3:31PM S18.00003 Rotation rate of rods in turbulent flow , SHIMA PARSA, Wesleyan University, ENRICO CALZAVARINI, Universite de Lille 1, Lille, France, FEDERICO TOSCHI, Eindhoven University of Technology, The Netherlands, GREG VOTH, Wesleyan University — We study the motion of single small rod-like particles in turbulent flow. The orientation and position of rods are measured experimentally using Lagrangian particle tracking with images from multiple cameras in a flow between two oscillating grids. We also have performed numerical simulations of rod motion in homogeneous isotropic turbulence. The probability distribution of the rotation rate of the rods has extended tails indicating rare events with large rotation rate. Rods rotation rate is determined by the velocity gradient of the flow, so measurements of the rotation rate provides indirect access to statistics of the velocity gradient of the flow. However, tracer rods preferentially sample the flow since their orientation becomes correlated with the local axes of the velocity gradient tensor. The result is that the typical rotation rate of rods is much smaller than it would be if they were randomly oriented. The numerical simulations allow us to quantify the preferential alignment, including its dependence on rod aspect ratio. This allows measurements of the second moment of the rod rotation rate to be used to measure the energy dissipation rate in the turbulent flow.

3:44PM S18.00004 Subgrid scale flux of heavy particle concentration in coarse grained Eulerian fields as deduced from DNS of 2D homogeneous isotropic turbulence , MARCELO CHAMECKI, Penn State University, J. CHRISTOS VASSILICOS, Imperial College London — Properties of preferential concentration of heavy particles in 2D homogeneous isotropic turbulence are investigated in an Eulerian framework. Particle motion is simulated in a Lagrangian framework, which is then used to derive the Eulerian particle concentration and velocity fields. Eulerian fields are coarse grained at several filter widths within the turbulence inertial subrange and statistics are analyzed. Focus is placed on the development of models to represent the subgrid scale (SGS) particle flux in large eddy simulation using the Eulerian equilibrium approach. It is argued that the SGS particle flux has two independent components that represent different physical processes: turbulent mixing due to SGS turbulence and clustering due to particle inertial response to SGS acceleration. These processes are completely independent and must be modeled separately. Phenomenological models based on centrifuging and the sweep-stick mechanisms are proposed for the inertial component.

3:57PM S18.00005 Lateral migration and orientation of ellipsoidal particles in Poiseuille flow , WENBIN MAO, ALEXANDER ALEXEEV, George W. Woodruff School of Mechanical Engineering, Georgia Institute of Technology — Inertial migration is widely used for separating microparticles in microfluidics. We use a hybrid numerical method that combine the lattice Boltzmann model and lattice spring model to investigate the dynamics of neutrally buoyant ellipsoidal particles in a Poiseuille flow with a finite Reynolds number. We find distinctive behaviors of particles depending on the particle shape, initial orientations and ratio of the particle size to channel width. Due to nonlinear flow effects, the particles equilibrate at off-center trajectories depending on particle geometry. Two possible equilibrium modes of motion are found. Particles either tumble with the main ellipsoid axis in the plane parallel to the flow direction or rotate along the main axis oriented perpendicular to the flow direction. We present a phase diagram showing the transitions between these two modes. The results indicate the possibility of particle separation by shape using microchannel flows.

4:10PM S18.00006 Effects of Polarization and Charging on the Deposition of Fine Particles on a Cylindrical Fiber: Simulation and Experiments[1] , SHUIQING LI, GUANQING LIU, MENGMENG YANG, Tsinghua University, JEFFREY MARSHALL, The University of Vermont — In this work, the particle deposition on a cylindrical fiber is investigated using a novel particle-level approach–discrete element method (DEM). The electrostatic effects, including both pre-charging and pre-polarization of fine particles, on the deposition patterns are discussed. Particularly, by this kind of method, a microscopic view on the depositions of neutral, polarized or charged particles is built, in which some basic finding are drawn. First, we find pre-polarization of fine particles enhances the deposition at nearly an order of magnitude; secondly, the charge of particle initially increases the deposition but finally inhibit it because of the repulsion between incident particles and deposited particles. A bench-scale set-up consisting of single fiber filtration system, particle charger, particle polarization unit and digital microscopy unit is built. The validation of DEM by experimental results is conducted.

[1]Supported by the National Natural Science Fundation of China (No.50976058).

4:23PM S18.00007 Intrinsic filtering errors of Lagrangian particle tracking in LES flow fields , CRISTIAN MARCHIOLI, University of Udine (Italy), FEDERICO BIANCO, SERGIO CHIBBARO, Pierre et Marie Curie University (France), MARIA VITTORIA SALVETTI, University of Pisa (Italy), ALFREDO SOLDATI, University of Udine (Italy) — Large-Eddy Simulations (LES) of two-phase turbulent flows exhibit quantitative differences in fluid and particle statistics if compared to Direct Numerical Simulations (DNS) which are considered here the exact reference case. Differences are due to filtering, a fundamental intrinsic feature of LES. In this paper, we quantify a lower bound for the filtering error using a DNS database of inertial particles dispersion in turbulent channel flow. Through ad hoc a-priori tests we single out the error purely due to filtering by removing error accumulation effects, which would lead to progressive divergence between DNS and LES particle trajectories. By applying filters of different type and width at varying particle inertia, we characterize the statistical properties of the filtering error. Results show that filtering error is stochastic and has a non-Gaussian distribution. In addition, the distribution of the filtering error depends strongly on the wall-normal coordinate being maximum in the buffer region. These findings provide insight on the effect of subgrid-scale velocity field on the force driving the particles, and establish the requirements which a LES model must satisfy to predict correctly particle segregation and preferential concentration.

4:36PM S18.00008 ABSTRACT WITHDRAWN —

4:49PM S18.00009 Lateral wall effects on the immersed pendulum dynamics and its interaction with a downstream target , FULING YANG, National Taiwan University, HONGSI CHEN, SOLID-LIQUID TWO-PHASE FLOW GROUP TEAM — The unsteady motion of a solid sphere in an incompressible viscous liquid and its interaction with solid boundaries has been examined and most existing knowledge concerns sphere motion in creeping flow regime or sphere collision dynamics. Much less investigated is when the solid sphere exhibits unsteady lateral motion relative to one or two planes. Thus, this work studies the dynamics of a fully immersed spherical pendulum when it moves midways between two lateral walls via systematic experiments. By changing the release angle and the liquid viscosity, a wide range of maximal particle Reynolds number, Re* from 3 to 295, were achieved without the presence of lateral walls. The same release was repeated with lateral walls imposed at various gaps, with a gap-to-diameter ratios W/D =1.2∼5.0, and the resulting maximum Re* were measured. We also estimated the coupling distance, Xc, when a downstream target sphere was set into motion by the impact sphere. If head-on collision was observed, the ratio of the relative sphere velocities after and before the collision were used to calculate the coefficient of restitution, e. The measured Re*, Xc, and e will be presented as a function of W/D along with some preliminary analysis.

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S20 Interfacial/Thin Film Instability VI 323

3:05PM S20.00001 Irrotational analysis of the early stages of break-up of a viscous drop in a high-speed gas stream , JUAN C. PADRINO, DANIEL D. JOSEPH, University of Minnesota - Department of Aerospace Engineering and Mechanics — The early stages of the break-up process of a liquid drop suddenly exposed to a high-speed gas stream behind a shock wave are considered. A linear analysis of the stability of the gas-liquid interface is conducted including the tangential component of the gas velocity near the interface and drop acceleration. The interfacial dynamics is thus governed by the combined mechanisms of Kelvin-Helmholtz and Rayleigh-Taylor instabilities. Visualizations of drop break-up by a gas stream at high Weber numbers reported in the literature reveal that in this regime instabilities driven by the shearing action of the gas play a role in the disintegration of the drop; this feature is central in developing the theory presented here. The dispersion relation for the growth rate and wave speed resulting from the stability analysis is written in terms of the density ratio, gas Weber and Reynolds numbers, and the liquid Ohnesorge number, which are typically used to specify an experimental run, and the Bond number, which contains the drop acceleration. Predictions from the stability analysis are discussed in the context of observations of experimental runs described in the literature for different values of the governing dimensionless parameters.

3:18PM S20.00002 The velocity of "large" viscous drops falling on a coated vertical fiber , JOHN HINCH, LIYAN YU, DAMTP, University of Cambridge — Kalliadasis & Chang (1994) showed within lubrication theory that if the coating is thicker than a critical value, drops accelerate and grow, whereas on thinner coatings, drops fall at a constant velocity. As the thickness of the coating increases to the critical value, the drops move faster and are bigger. We revisit their asymptotic analysis of these large drops.

3:31PM S20.00003 Solidification of molten metallic foams[1] , PETER STEWART, MICHAEL DAVIS, STEPHEN DAVIS, Northwestern University — High-porosity metallic solids can be formed by solidification of the corresponding molten gas-liquid foam. However, molten metallic foams are thermally and dynamically unstable, so in the absence of solidification the thin liquid films drain rapidly toward the bubble vertices and eventually become unstable to interfacial instabilities, leading to film rupture and bubble coalescence. To explore the competition between coarsening and freezing we have constructed a large-scale network model to describe the dynamics and stability of a planar foam with low liquid fraction, incorporating a coupling between pressure and volume in the gas bubbles, surface tension forces on the gas- liquid interfaces, draining flow in the films, a criterion for film rupture, thermal fluctuations and a solidification front. Initially, the foam is arranged in a regular array of approximately polygonal bubbles, held at a uniform temperature above the melting point of the material. The walls of the container are then cooled to a temperature well below the melting point, driving a solidification front inwards; numerical simulations of the model predict the structure of the resulting porous metal solid.

[1]National Science Foundation Grant Nos. CMMI-0826703 (PSS,SHD) and DMS-0636574 (MJD)

3:44PM S20.00004 Biaxial extensional motion of an inertially driven liquid disk[1] , LINDA SMOLKA, Bucknell University — We derive a time-dependent exact solution of the free surface problem for the Navier-Stokes equations that describes the biaxial extensional motion of a viscous disk driven by inertia. The linear stability of the exact solution to axisymmetric and two-dimensional perturbations is examined in the inviscid limit within the framework of the long-wave approximation. Both transient growth and long-time asymptotic stability are considered.

[1]NSF Grant No. DMS-0707755

3:57PM S20.00005 Stabilizing the Interface in a Saffman-Taylor Problem by Heating , RANGA NARAYANAN, ERDEM UGUZ, LEWIS JOHNS, University of Florida, UNIVERSITY OF FLORIDA TEAM — The interface in a Saffman Taylor problem can be stabilized to perturbations of *any wave length* by simply heating from above. The same is true for the Rayleigh Taylor instability. We present simple formulas for estimating the temperature difference required to do this and find that more reasonable temperature differences obtain in the Saffman-Taylor problem because the temperature dependence of the viscosity is ordinarily much stronger than the temperature dependence of the density.

4:10PM S20.00006 Electrohydrodynamic instabilities of biomimetic bilayer membranes , JACOPO SEIWERT, PETIA VLAHOVSKA, Brown University — A DC electric pulse can destabilize a fluid bilayer membrane, causing a transient bending of the interface [1]. The membrane deformation relaxes as the membrane capacitor charges. We investigate the possibility of using AC electric fields and charged lipids to control the instability. In AC fields, the dynamics of a charge-free membrane depends on the relative magnitudes of the inverse of the field frequency and the capacitor charging time. At low frequencies, where the capacitor is fully charged and draws no current, the membrane is stable. As the frequency increases and the capacitor becomes short-circuited, the membrane behaves as a simple interface separating leaky dielectric fluids and can become unstable depending on the fluid electric properties (conductivities and dielectric constants). In the presence of native charge, the charge redistribution leads to even more complex membrane dynamics.

[1] Schwalbe et al. Physics of Fluids 23: 041701 (2011)

4:23PM S20.00007 The effect of permeability gradients on immiscible displacement in Hele-Shaw flows[1] , TALAL AL-HOUSSEINY, Chemical and Biological Engineering Department, Princeton University, PEICHUN TSAI, ZHONG ZHENG, HOWARD STONE, Mechanical and Aerospace Engineering Department, Princeton University, COMPLEX FLUIDS GROUP TEAM — In heterogeneous media, it is well known that when a fluid of high viscosity displaces a less viscous fluid, the interface can still be unstable and exhibit finger-like patterns due to capillary fingering. Motivated by porous media flows in natural geological formations, we consider homogeneous displacement in a Hele-Shaw cell subjected to a permeability gradient. The permeability gradient is introduced by linearly varying the Hele-Shaw cell depth. We study how capillary forces can affect interfacial stability in the presence of the gradient via linear stability analysis. Depending on the system, we find that surface tension can either have a stabilizing or a destabilizing role. We report the emergence of an important dimensionless parameter–the ratio of the permeability gradient to the capillary number–that determines the stability of the interface along with the well-studied viscosity ratio. Experiments testing the theoretical findings will also be presented.

[1]This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE-0646086

4:36PM S20.00008 Contact-line dynamics, bifurcation and bi-stability of droplets driven by thermal gradients , JOSHUA BOSTWICK, MICHAEL SHEARER, KAREN DANIELS, North Carolina State University — We consider the quasi-static spreading of a sessile drop on a radially-heated, partially-wetting substrate. In general, for a non-thermally simple fluid, this type of heating can generate both axial and radial temperature gradients along the drop interface, which produce thermo-capillary forces. These generate flows that affect the spreading process through the contact-line dynamics. Our model employs lubrication theory together with a constitutive law that relates contact-angle to contact-line speed. We identify parameter regimes in which the droplets continue to spread indefinitely, and compute spreading exponents. In regions of parameter space in which the droplets converge to an equilibrium shape, we show that competition between surface chemistry (wetting conditions) and flow induced by thermal gradients can give rise to bi-stability. The path to equilibrium is then typically complex and the droplet may evolve through a number of intermediate states, such as a capillary ridge or a shape whose bulk and contact-line regions have essentially de-coupled.

4:49PM S20.00009 Stability of Liquid Rivulets on Liquid Substrate , COLIN CERRETANI, SHO TAKATORI, CLAYTON RADKE, University of California, Berkeley — The human tear-film lipid layer is a thin (100nm) oily film on water. Such films are unstable and dewet into lenses surrounded by a monolayer (Harkins, 1941). Dewetting has four stages: initial rupture, hole growth, hole coalescence, and retraction into lenses. The human lipid layer is shown to behave similarly. Brochard-Wyart has addressed the first two stages (1993); here we focus on the third. As adjacent holes grow into each other, the oil between them takes the shape of a long, thin rivulet with a lens cross-section. Eventually this rivulet undergoes an instability and the holes coalesce. We perform a linear stability analysis on a thin symmetric lens rivulet on a horizontal liquid substrate at low Bond number (Davis, 1980; Schiaffino, 1997), accounting for the first time for the liquid-substrate hydrodynamics. Analytical expressions are derived for the wavelength and breakup times associated with the maximum growth rate of the instability for multiple substrate flow conditions. We show that for negligible thin-film forces, a liquid rivulet on an immiscible liquid substrate is unstable at a critical wavelength disturbance. The instability growth rate varies by orders of magnitude depending on the lens contact angle.

5:02PM S20.00010 Contact line instability of a liquid rivulet partially wetting an inclined plane[1] , A.G. GONZALEZ, J.A. DIEZ, IFAS, Universidad Nac. del Centro de la Prov. de Buenos Aires, Tandil, Argentina, L. KONDIC, Dept. of Mathematical Sciences, Center for Applied Mathematics and Statistics, New Jersey Inst. of Technology, Newark, NJ — We analyze the stability of a liquid rivulet of cross section, A, positioned across a plane with inclination angle, α. The liquid partially wets the substrate with a static contact angle, θ_0, when the substrate is horizontal. The contact line stability is studied using the lubrication approximation and with a slip model. Both normal and parallel components of gravity are included A static solution exists for small α's and its linear stability is considered. We use an pseudo-spectral Chebyshev method with a combination of basis functions that automatically satisfies the conditions at the contact lines. We analyze the effects of A, θ_0 and α on the predictions of the model, such as stability regions, the maximum growth rate and the behavior of most unstable perturbation. Experiments with silicone oils spreading on a coated glass substrate are considered for a number of different fluid volumes and α's. We find a good agreement between the wavelength of maximum growth predicted by the model and the experimental average distance between drops.

[1]Supported by ANPCyT-PICT2498/06, CONICET(Argentina) and NSF Grant DMS-0908158.

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S21 Vortex Dynamics VIII 324-325

3:05PM S21.00001 Particle-vortex interactions in quantum turbulence[1] , ENRICO FONDA, Università di Trieste - University of Maryland, KATEPALLI R. SREENIVASAN, New York University, DANIEL P. LATHROP, University of Maryland — We study quantum turbulence in a thermal counterflow of He II, using sub-micron and micron-sized frozen particles. For low heat fluxes, these particles can either trace the motion of the normal component or track quantized vortices when they get trapped on their cores. For high heat fluxes the increased number of particle-vortex interactions and scattering events result in a different state in which the particles track neither the vortices nor the normal component. Analyzing the trajectories of tracers of different size and density for a wide range of heat fluxes, we investigate the particle-vortex interaction mechanism. The results are used to test analytical and numerical models, and explain the discrepancy between two previous counterflow experiments.

[1]This research was supported by the NSF-DMR.

3:18PM S21.00002 Effects of a flexible margin on Robojelly vortex structures , ALEX VILLANUEVA, KELLEY STEWART, PAVLOS VLACHOS, SHASHANK PRIYA, Department of Mechanical Engineering, Virginia Tech — An Unmanned Underwater Vehicle (UUV) inspired by jellyfish morphology and propulsion mechanism, termed "Robojelly," was used to analyze the effects of the flexible margin on jellyfish propulsion. The natural animal has a bell section which deforms at a different phase then the rest of the bell. This lagging section, referred to as flexible margin or flap, is delimited by the bell margin and an inflexion point. The flap was replicated on the robotic vehicle by a flexible passive material to conduct a systematic parametric study. In a preliminary experiment, Robojelly was tested without a flap and with a flap. This revealed a thrust increase over an order of magnitude. We hypothesize that the length of this passive flap affects the vortex ring circulation strength of the jellyfish which can lead to higher efficiency and thrust. Velocity field measurements were performed using planar Time Resolved Digital Particle Image Velocimetry (TRDPIV) to analyze the change in vortex structures as a function of flap length. The robot input parameters stayed constant over the different configurations tested thus maintaining a near constant power consumption. Results clearly demonstrate that the flap plays an important role in the propulsion mechanism of Robojelly and provides an anatomical understanding of natural jellyfish.

3:31PM S21.00003 3D Reconstruction of the Vortex in a Human Right Ventricle Model using High Speed PIV , ARASH KHERADVAR, AHMAD FALAHATPISHEH, University of California Irvine — This work aims to characterize the formation process and translation of the vortex, which forms along with the trans-tricuspid jet in a realistic model of a human right ventricle (RV). A clear model of the RV made of silicone rubber was carefully casted in real size from echocardiographic data of an adult human heart. The RV model was used in our heart pulsed-flow simulator at KLAB at UCI to perform experiments. Bioprosthetic heart valves in appropriate sizes were used at tricuspid and pulmonary positions. Multi-planar high-speed PIV was performed to capture and reconstruct the 3D flow field with a 1-millisecond time gap between each two velocity frames. λ_2 iso-surfaces were used to illustrate the evolution of vortex cores. The highly asymmetric shape of the RV chamber results in a complex 3D trans-tricuspid vortex that forms and translates toward right ventricular outflow tract, and finally departs RV from pulmonary valve. Through this study, -for the first time-the formation, evolution and pathway of the RV vortex have been characterized *in vitro*.

3:44PM S21.00004 Experimental study of asymmetric heart valve prototype , M. VUKICEVIC, University of Trieste , S. FORTINI, University of Roma, G. QUERZOLI, University of Cagliari, A. CENEDESE, University of Roma, G. PEDRIZZETTI, University of Trieste — The mechanical heart valves (MHVs) are extremely important medical devices, commonly used for diseased heart valves replacement. Despite the long term of use and constant design refinement, the MHVs are very far from ideal and their performance is very diverse from that of the native ones. It has been approved that small variations in geometry of valvular leaflets influence the significant change in the intraventricular vortical flow, known as one of the most important factors for the overall functionality of the heart. We have experimentally examined the home-made heart valve prototypes, exclusively modeled for the mitral valve replacement. The performance and energetic properties of the prototypes have been compared with those in the presence of standard MHVs. The analysis was based on the testing of intraventricular fluid dynamics, usually missing criteria for the quality of the valve performance. It has been shown that the asymmetric prototype, with unequal leaflets and D-shaped orifice produces flow patterns and energetic properties close to those found in the healthy subjects. Thus, the break of symmetry in the standard bi-leaflet MHV prosthesis, at least from the fluid dynamics point of view, is worthwhile to be considered for the design of MHVs for the mitral valve replacement.

3:57PM S21.00005 The whale footprint , ADRIEN BENUSIGLIO, CHRISTOPHE CLANET, LadHyx, Ecole Polytechnique — In their wake whales leave patches of very calm water, with few waves, that can last for several minutes known as whale footprints. The same phenomenon of damping of waves appears near pears of bridges, in the turbulent wake of ships or in rivers re-emergences, thus it appears to be the result of the interaction of a turbulent flow with surface waves. To understand this phenomenon we study the interaction of a single vortex ring with surface waves. We investigate the influence of the size and circulation of the vortex ring. We measure the time of interaction and the size of the resulting patch of calm water. We then verify if our model can explain the whale footprint.

4:10PM S21.00006 Effect of slip on vortex dynamics and forcing of a superhydrophobic cylinder , PRANESH MURALIDHAR, ROBERT DANIELLO, NANGELIE FERRER, JONATHAN ROTHSTEIN — Superhydrophobic surfaces have been shown to produce significant drag reduction for both laminar and turbulent flows of water through large and small-scale channels. In this presentation, a series of experiments will be presented which investigate the effect of superhydrophobic-induced slip on the vortex dynamics in the wake of a cylinder and the change in the drag and lift forces thereof. In these experiments, circular cylinders are coated with a series of superhydrophobic surfaces fabricated from PDMS with well-defined micron-sized patterns of surface roughness. Using force measurements and PIV (Particle Image Velocimetry), we will show that these surfaces have a noticeable effect on the drag/lift and vortex dynamics of cylinders. When compared to a smooth, no-slip cylinder, we will show that the lift/drag and the amount of raw vorticity that is shed in the wake of the superhydrophobic cylinder decreases. In addition, we will show that the forcing is sensitive to changes of feature spacing, size and orientation.

4:23PM S21.00007 VIV of cylinders with superhydrophobic coating , ROBERT DANIELLO, JONATHAN ROTHSTEIN, University of Massachusetts, Amherst — For several years, superhydrophobic surfaces which are chemically hydrophobic with micron or nanometer scale surface features have been considered for their ability to reduce drag in microfluidic devices. More recently it has been demonstrated that superhydrophobic surfaces reduce friction coefficient in turbulent flows as well. In this talk, we will consider the effect of superhydrophobic surfaces on vortex induced vibrations of an elastically mounted circular cylinder. Effects on frequency and amplitude will be considered and particle image velocimetry will be used to examine vortices shed from the superhydrophobically coated moving cylinders. Several unique superhydrophobic microstructures will be used in the Cassie and Wenzel states to demonstrate the effect of microstructure and preferential slip direction on the oscillations of the cylinder.

4:36PM S21.00008 Evolution and scaling of a start-up vortex from a two-dimensional shear layer , POORIA SATTARI, DAVID RIVAL, ROBERT MARTINUZZI, University of Calgary, CAMERON TROPEA, Technische Universitaet Darmstadt — The formation process of a vortex due to the impulsive start of a shear layer plays an important role in a large number of industrial and biological systems. From the point of view of aerodynamic efficiency and momentum transfer, reaching higher levels of circulation prior to the release of the vortex can often be advantageous. For this reason the determination of a possible limit to vortex size would be an important step forward in our understanding of the vortex pinch-off process. In the present work, the evolution of an isolated line vortex generated by a starting two-dimensional jet is studied experimentally with the aid of time-resolved PIV. Unlike previous studies, this current experiment is not restricted by any interactions with other vortical structures or walls that potentially limit vortex size. By tracking the circulation, size and position of the vortex relative to the shear layer, we were able to identify the limit to vortex growth and therefore better understand the physical pinch-off process. Furthermore, in the absence of inherent velocity and length scales, alternative parameters for the scaling of circulation based on shear-layer characteristics are proposed instead.

4:49PM S21.00009 Inverse cascades in an experimental turbulent von Kármán swirling flow , MIGUEL LOPEZ, JAVIER BURGUETE, UNAV, MHD TEAM — An experimental study of the energy spectrum in a von Kármán swirling flow is presented. For the experiments we use water stirred in a closed cylindrical cavity placed in a fully developed turbulent regime ($Re \sim 10^5$). The three components of the velocity field (v_R, v_θ, v_z) can be measured with a 1D laser doppler velocimetry, one component at a time. Different time scales can be identified in the fourier space spliting the spectrum in different cascades with different slopes. These timescales are a consequence of instabilities in the mean flow. Two different inverse cascades (slopes -2 and $-1/3$) appear in some regions of the flow whose origin is related to the dynamics of coherent structures. The existence of these times scales can affect the statistical characterization of the turbulent flow.

5:02PM S21.00010 Buoyancy-Induced Columnar Vortices , MARK W. SIMPSON, ARI GLEZER, Georgia Institute of Technology — Large-scale inherent instability of a thermally stratified air layer that is heated from below by a thermal ground plane in a meter-scale laboratory facility is exploited for the deliberate formation of intense columnar vortices. In nature, such buoyancy-driven vortices ("dust devils") occur spontaneously, with core diameters of 1-50 m at the surface, heights up to one kilometer, and with induced air flow of considerable angular and linear momentum. The laboratory experiments have demonstrated the nucleation and sustainment of strong, buoyancy-driven vortices that are anchored and stabilized using an azimuthal array of flow vanes while the temperature of the thermal ground plane is maintained using a resistive heater and a controllable power source. The present investigation focuses on the fundamental mechanisms of the formation, evolution, and dynamics of the columnar vortex using stereo-PIV. It is also shown that the strength and scaling of theses vortices can be significantly altered by adjustment of the flow vanes and the amount of global sensible heat absorbed by the air flow, or the "buoyancy flux". The characteristics of the anchored vortices can also be altered by small modifications of the thermal ground plane that affect the flux and distribution of the ground plane vorticity that forms the core of the vortex.

Tuesday, November 22, 2011 3:05PM - 4:10PM –

Session S23 Flow Control VII 326

3:05PM S23.00001 Synthetic Jets Flow Control on a vertical stabilizer , NICHOLAS RATHAY, MATTHEW BOUCHER, MICHAEL AMITAY, Rensselaer Polytechnic Institute — The vertical stabilizer on most commercial transport aircraft is much larger than required for stability and control. The tail is significantly oversized in order to maintain controllability in the event of asymmetric engine failure and meet flying qualities requirements related to dynamic motion. Using aerodynamic flow control techniques, it may be possible to reduce the size of the tail while maintaining similar control authority during inclement flight conditions. Reducing the size of the tail decreases the weight and the drag of the airplane, which results in considerable savings in fuel costs. In this work, it is shown that synthetic jet (zero-net-mass-flux) actuators are capable of reattaching the separated flow on the rudder and augmenting the performance of the stabilizer. Experiments were conducted in an open-return wind tunnel on a 1/25th scale model of a vertical stabilizer and a partial fuselage section. The surface pressure, aerodynamic loads and data acquired with a Stereo PIV system were used to investigate the effectiveness of this technology as well as provide a more detailed analysis of the flowfield and showed that the synthetic jets are capable of augmenting the side-force by up to 20%.

3:18PM S23.00002 Interactions of a Propeller with a Circumferentially-Varying Flow , JOHN FARNSWORTH, US Air Force Academy, MICHAEL AMITAY, Rensselaer Polytechnic Institute, DAVID BEAL, STEHPEN HUYER, Naval Undersea Warfare Center, Newport — The interactions of a circumferentially varying stator cascade and a downstream fixed pitch propeller were investigated experimentally in a water tunnel using the Stereoscopic PIV technique. A cyclic distribution of the stators' deflections resulted in non-axisymmetric distributions of the flow field downstream of the stator array. The stator distribution alone produced a significant side force that increased linearly with stator pitch amplitude. When a propeller was incorporated downstream from the cyclic cascade the side force from the stator cascade was reduced, but a small normal force and pitching moment were created. The generation of these secondary forces and moments can be related to the redistribution of the tangential flow from the cyclic cascade into the axial direction by the retreating and advancing blade states of the fixed pitch propeller. The Q criterion was utilized with measurements phase-locked to the propeller frequency to visualize the interactions of the vortical structures in the wake of the combined stator-propeller system.

3:31PM S23.00003 On the Introduction of Unsteady Streaks into a Blasius Boundary Layer Using Dynamically Actuated Roughness[1] , KYLE BADE, AHMED NAGUIB, Michigan State University, Mechanical Engineering, PHILIPPE LAVOIE, University of Toronto, Institute for Aerospace Studies — Observations of the spatio-temporal growth of streamwise elongated streaks emanating from a cylindrical roughness element undergoing dynamic actuation into/out of a Blasius boundary layer are presented. Hot-wire measurements provide 2D maps of the disturbance velocity of these streaks. Modal decomposition methods are performed on these maps to evaluate the nature of the temporal growth of the streaks. For this analysis, various roughness element actuation heights, velocities, and accelerations are examined in order to identify the "dynamic-roughness" actuation parameters range for which transiently growing streaks can be produced while avoiding the introduction of T-S wave packets and/or non-linearly-growing disturbances. The establishment of such streaks with 5-10% disturbance magnitude will provide the basis for an experimental platform, and help develop efficient models for feedback bypass transition control in an ongoing study in collaboration with Princeton University.

[1]Funded by NSF grant number: CMMI 0932546

3:44PM S23.00004 Active Control of Vortex Induced Vibrations of a Tethered Sphere in a Uniform Air Flow , RENE VAN HOUT, Technion-Israel Institute of Technology, DAVID GREENBLATT, AMIT ZVI KATZ, Technion - Institute of Technology — VIV of two heavy tethered spheres ($D = 40$ mm, $m^* = m_{sphere}/\rho_f V_{sphere} = 21$ and 67, $L^* = L/D = 2.50$) were studied in a wind tunnel under uniform free stream velocities up to $U^* = U/f_n D = 15.9$, with and without acoustic control. Control was achieved using two speakers mounted on either side of the spheres and driven in-phase at $f=$ 35Hz ($f^* = 22.3$). In the non-controlled case, the bifurcation map of transverse sphere oscillation amplitude, A_y, showed stationary motion as well as periodic and non-stationary oscillations with increasing U^*. For $m^* = 21$, $A_y^{\max}$ was about twice as large as for $m^* = 67$. Acoustic control dampened $A_y^{\max}$ in the periodic region ($m^* = 67$) and increased $A_y^{\max}$ in the non-stationary region for both spheres. Sphere boundary layer dynamics in the three different bifurcation regions were studied using time resolved PIV with a horizontal laser sheet positioned at the center of the sphere. The field of view was 55x55 mm^2 containing one quarter of the sphere. Results will be presented on the vortex dynamics near the sphere's surface with and without acoustic control.

3:57PM S23.00005 Aero-Effected Flight Control Using Distributed Active Bleed[1] , JOHN M. KEARNEY, ARI GLEZER, Georgia Institute of Technology — The aerodynamic effects of large-area air bleed that is driven through surface openings in a lifting airfoil by pressure differences are investigated in wind tunnel experiments. The bleed is actively regulated by arrays of addressable integrated louvers, and the time-dependent interactions between the bleed and cross flows alter the apparent aerodynamic shape of the lifting surface, and, consequently, the distributions of aerodynamic forces and moments. The 2-D Clark-Y wind tunnel model is oscillating in pitch, and the dynamic changes in the time-dependent forces and moment are measured over a wide range of angles of attack from pre- to post-stall using integrated load cells. Induced changes in surface vorticity concentrations are measured using PIV with emphasis on the effects of leading edge bleed at high angles of attack on the evolution of the dynamic stall vorticity concentrations.

[1]Supported by AFOSR

Tuesday, November 22, 2011 3:05PM - 4:36PM –

Session S24 Convection and Buoyancy Driven Flows III 327

3:05PM S24.00001 Simulation of a submarine hydrothermal convection using a numerical model for a supercritical water flow[1] , SATOKO KOMURASAKI, Nihon University — Hydrothermal convective flows derive from geothermally heated water issued from hydrothermal vents. They are found even in deep oceans of depth over 2,000 meters where the hydrostatic pressure of water is over 200 atmospheres, and temperature of heated water is occasionally more than 300°C. Under these conditions, water emerging from these vents can be supercritical. In reality, it is quickly cooled by ambient water of temperature approximately 2°C and becomes liquid. It is also important to investigate of advection and diffusion of materials which are issued from hydrothermal vents accompanying heated water, in order to deepen understanding of the oceanic crust that possesses abundant resources. A flow simulation is carried out using a numerical model for a supercritical water flow where it is assumed that the pressure is high and almost constant, and the density changes only by temperature. In the computation, the modified compressible Navier-Stokes equations are solved by a method similar to an approach used in solving the incompressible equations. Computation is done employing finite difference method in 2D. Simulated flow fields are displayed using a suitable visualization technique and compared with the results of simulations based on the Boussinesq approximation.

[1]This work was partially supported by KAKENHI (22740261)

3:18PM S24.00002 Convective Thermal Transport in Near-Critical Fluids , NUSAIR HASAN, BAKHTIER FAROUK, Department of Mechanical Engineering and Mechanics, Drexel University — Convective thermal transport in a pure fluid (carbon dioxide) near its gas-liquid critical point is investigated using a two-dimensional numerical model. A square enclosure is considered with bottom heating. The model considers the strong variable property effects (functions of both temperature and pressure) near the critical point, including the bulk viscosity variations. Although thermal diffusivity approaches zero near the critical point, the divergence of thermo-physical properties near the critical point gives rise to large Grashof-number flows even for very small temperature differences. The convective heat transfer coefficient near the critical point is investigated and a correlation for the Nusselt number is developed as a function of the Rayleigh number and the ratio $(p_m T_m / p_{cr} T_{cr})$ where the subscripts "m" and "cr" denote the mean and critical values respectively. As the critically diverging bulk viscosity plays a significant role on the transport processes near the critical point, effect of bulk viscosity on the flow field and heat transport induced by buoyancy in near-critical fluids is also investigated.

3:31PM S24.00003 Thermal-hydraulic behavior of Sc-C02 in a horizontal circular straight tube[1] , KATSUYOSHI TANIMIZU, Postdoctoral Research Associate, REZA SADR, DAVESH RANJAN, Assistant Professor — Fluids above critical pressure have been practically utilized for 60 years in many applications and their use and interest is still increasing in many areas, especially power generation industries and chemical industries. Above critical pressure, very rapid changes in thermophysical properties take place near the pseudocritical temperature. In this region, the fluid transforms from liquid-like to gas-like behavior when the fluid temperature rises up and passes through the pseudocritical temperature. This allows enormous potential for energy transfer, but also alters the turbulent flow due to changes in the turbulent shear stress brought about by acceleration and buoyancy effects. However, we have not fully understood their dynamic behaviors such as turbulence yet. A supercritical CO_2 testing loop has been built at Texas A&M University at Qatar to perform heat transfer and pressure drop measurements and investigate the thermo-physical and dynamic characteristics of supercritical carbon dioxide flow. The results of heat transfer measurements in a super critical fluid conducted in a horizontal pipe are reported and discussed here.

[1]Supported by QNRF

3:44PM S24.00004 Finite-sample-width effects on chimney formation in mushy-layer convection , ANDREW WELLS, JIN-QIANG ZHONG, ANTHONY FRAGOSO, JOHN WETTLAUFER, Yale University — The rapid solidification of a binary alloy leads to the formation of a chemically reactive porous medium, or mushy layer, within which convection and dissolution can generate *chimneys*: drainage channels devoid of solid. We combine experiment with numerical simulation to explore the influence of a finite system size on the dynamics and stability of the convective flow. For a variety of steady growth rates and cell widths, we determine conditions for chimney formation during experimental solidification of aqueous ammonium chloride in a Hele-Shaw cell. The results are compared to a two-dimensional numerical model of mushy-layer convection, providing insight into the prediction of defect formation in industrial casting processes.

3:57PM S24.00005 A simple one-dimensional model of brine fluxes from sea-ice , GRAE WORSTER, DAVID REES JONES, DAMTP, University of Cambridge — Over the winter growth season, brine within the interstices of sea-ice drains into the ocean owing predominantly to convection within the lower parts of the ice, a small-scale process difficult to model on a regional scale. Traditionally, salt loss from sea-ice has been determined using a combination of historical data and an empirical segregation law applied at the ice-ocean interface. By contrast, we apply a simple relationship between the strength of internal, small-scale convection and a mush Rayleigh number to develop a predictive, one-dimensional model of sea ice, applicable on a regional scale. This model solves the full mushy layer equations for heat and salt conservation within the ice, using a dynamically determined term for the convective transport of salt and heat. Numerical results from this model are presented, predicting the internal structure of the ice and also net salt and heat fluxes from the ice. Prospects for experimental validation and further possibilities for model development are discussed.

4:10PM S24.00006 Fluxes through steady-state chimneys in binary alloy solidification , DAVID REES JONES, GRAE WORSTER, DAMTP, University of Cambridge — Solute fluxes from mushy layers in solidifying binary alloys occur predominantly by convection through nearly vertical, straight-sided chimneys. We develop a simple, theoretical study using a patchwork of simplifying approximations that capture the essential physics while rendering the problem analytically tractable. Away from the chimney, the temperature field is horizontally uniform and vertically linear. Near the chimney, a similarity solution with linear vertical structure is found. The lubrication approximation to Stokes flow applies within the chimney itself, and the chimney is proved to be straight-sided. The melt is treated through a boundary-layer parameterization of the heat flux. A linear flux-Rayleigh number relationship is proved analytically in planar geometry, based on a criterion on chimney spacing that optimises the buoyancy flux. In three dimensions, the crucial importance of drainage area is demonstrated, and an approximately linear flux-Rayleigh number relationship is found. This constitutes a simple, physically motivated representation of solute flux, that might be used cheaply in global sea-ice models.

4:23PM S24.00007 Mushy layer dynamics in micro and hyper gravity[1] , JOSEPH O'ROURKE, A.J. ELDORADO RIGGS, CHARLOTTE GUERTLER, PEARSON MILLER, CATHERINE PADHI, MADELINE POPELKA, ALLISON WEST, JIN-QIANG ZHONG, JOHN WETTLAUFER, Yale University — We describe the results of experiments on mushy layers grown from aqueous ammonium chloride solution in normal, micro, and hyper gravity environments during parabolic flight. In the fully developed chimney state, the chimney plume dynamics differ strikingly when conditions change from microgravity to hypergravity. In microgravity, we find fully arrested plume motion and suppressed varicosity. As gravity transcends Earth conditions, we observe a host of phenomena, ranging from arched plumes that undergo forced Rayleigh-Taylor instabilities to in-phase multiple plume oscillatory behavior. For the same initial solute concentrations and fixed chill temperatures, we find that, in runs of over 130 minutes, the averaged effects of hypergravity result in suppressed growth of the mushy layers, a phenomenon caused by a net enhancement of convective heat transport from the liquid to the mushy layers. These behaviors are placed in the context of the theory of convecting mushy layers as studied under normal laboratory conditions.

[1]Support from CT Space Grant, Yale University, and NASA JSC's Reduced Gravity Student Flight Opportunities Program.

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S26 Surface Tension IV

3:05PM S26.00001 Immersed parallel plates: from repulsion to attraction , PAUL CONCUS, Lawrence Berkeley National Laboratory — This mathematical study concerns the behavior of parallel vertical plates of infinite extent and of possibly differing materials dipped into an infinite liquid reservoir. For this classical problem varying modes of behavior can occur, depending on plate separation and contact angles. Under specific conditions the plates will repel each other when sufficiently far apart, with change to large attraction, which can be abrupt, when they are moved closer together. As separation is decreased to a critical value, the liquid/vapor interfaces pass through a family tending to a certain, explicitly known interface yielding zero force. For the initial stages of this procedure computational results depict a striking behavior, that the interfaces in the sequence can actually move away from the limiting interface, not toward it.

3:18PM S26.00002 Direct Observation of Hierarchical Contact Line Depinning , ADAM PAXSON, KRIPA VARANASI, Massachusetts Institute of Technology — We report a technique for observing the dynamic behavior of a liquid contact line at high magnification using environmental scanning electron microscopy. We find that on a superhydrophobic surface consisting of an array micropillars, the receding contact line exhibits discrete hierarchical de-pinning events. As the macroscopic contact line recedes across the pillars, a capillary bridge is formed and displays a local microscopic contact angle that is equivalent to the macroscale contact angle observed on a flat surface of the same composition. By considering the line density of the microscale features and the pinning strength of each of those features, we relate the macroscopic contact angle and adhesion to the multiscale hierarchical roughness. This mechanism helps to explain the necessity for multiple length scales exhibited by lotus leaves and other superhydrophobic surfaces.

3:31PM S26.00003 Large Scale Dynamics and Partial Coalescence of Soap Bubbles[1] , DAVID MARTIN, FRANCOIS BLANCHETTE, UC Merced — We present simulations of the large scale dynamics of soap films. Our model uses a thin, axisymmetric interface, with finite thickness and mass. The interface is treated as a two dimensional front, and we track the local thickness and surfactant concentration. We present results on the coalescence of soap bubbles of different sizes. Conditions are found under which this coalescence is only partial.

[1]We acknowledge support from NSF grant DMS 0808129

3:44PM S26.00004 ABSTRACT MOVED TO L20.00005 —

3:57PM S26.00005 Finite Bond number effects in the dripping nozzle , SUHWAN CHOI, THOMAS WARD, North Carolina State University — The pressure-drop in a dripping nozzle generated by hydrostatic pressure and surface tension is studied by both theory and experiments. As the hydrostatic pressure is varied in each experiment, the measured pressure-drop can change sign depending on a inner tip diameter, Bond number, angle of valve, and viscosity. The viscous fluids used in the experiments are glycerol-water mixtures with viscosities ranging from 0.176 to 1.09 Pa s and silicone oil having a viscosity of 0.418 Pa s and higher Bond number compared with the mixtures. The pendant drop growth is taken by CCD camera to make observations. The images are examined for the radius of the drop as a function of time and compared with the derived momentum equation which include Bond number effects. We propose a process to use the analysis for estimating the surface tension without the need to measure the interface curvature.

4:10PM S26.00006 Capillary rise in cellulose sponges , JUNGCHUL KIM, HO-YOUNG KIM, Seoul National University, L. MAHADEVAN, Harvard University — A cellulose sponge, commonly used for clean-up jobs, can absorb and hold a significant amount of water within its pores, whose size ranges from micrometers to millimeters. We investigate the dynamics of capillary rise of water in the sponge using a combination of experiment and theory. We find that the rate of the capillary rise is significantly lower than Washburn's rule that assumes the sponge as a row of adjoined pores and the liquid flow to be driven by the Laplace pressure. We introduce a novel theory to model the flow in the hygroscopic porous media by combining Darcy's law based on the moisture concentration and the modified Young-Laplace equation. The scaling law constructed through this work agrees well with the experimental results.

4:23PM S26.00007 Coupling between meniscus dynamics and corner flow near a junction , LINGGUO DU, HUGUES BODIGUEL, ANNIE COLIN, LOF, CNRS UMR 5258, univ. Bordeaux 1, Rhodia — The movement of a liquid-liquid meniscus in a circular tube at small capillary numbers is a rather simple problem since it follows Poiseuille's law with an effective pressure that incorporates the capillary pressure. However, for a tube of arbitrary cross-section exhibiting sharp corners, capillarity requires that the wetting liquid remains along the corners. These corner films may flow. In this work we experimentally investigate the role of these corner flows on the dynamics of a liquid-liquid meniscus, in total wetting conditions, by taking advantage of a microfluidic PDMS cross-junction. Our results show that there is a strong coupling between the corner flows and the displacement of meniscus, when it lies in the vicinity of the junction (up to 10 times the channel width 50μm). Part of the observations is accounted for by a model based on a network of adaptive hydrodynamic resistances. However, when the size of the corner film is high, a more detailed description of the interface shape is required to account for the observations.

4:36PM S26.00008 Microliquid prism actuated by electrowetting , DUCK-GYU LEE, JAEBUM PARK, HO-YOUNG KIM, Seoul National University — A microliquid prism is a microchannel filled with two immiscible liquids whose interface acts as a refractive surface. To steer a beam to construct optical images, the interface profile or the contact angle is modulated through electrowetting. The effective actuation of the liquid prism critically depends on the understanding of liquid flows around the moving interface. We visualize the flow field near the moving contact line formed by water, silicone oil and a glass microchannel using PIV (particle image velocimetry). The flow, which is believed to be visualized for the first time to the authors' knowledge, is clearly distinguished from that in the classical microliquid slugs in that the fluid motion near the electrowetting-driven contact line is localized in the vicinity of the contact line. We also provide a theoretical model to predict the temporal evolution of the interface profile as a function of the liquid properties and the dependency of the contact angle on voltage.

4:49PM S26.00009 Dynamic surface tension effects from molecular dynamics simulations[1] , ALEX LUKYANOV, ALEXEI LIKHTMAN, University of Reading — We will present results of our recent large scale molecular dynamics simulations of dynamic surface tension behaviour in the case of a liquid-gas interface. We will demonstrate the mechanism of surface tension relaxation from a non-equilibrium state in several representative cases: long-chain flexible molecules with Lennard-Jones beads connected by FENE springs and binary Lennard-Jones mixtures (Kob-Andersen model). The methodology of the surface tension evaluation has been successfully tested against the Laplace law in all the cases.

[1]This work has been supported by EPSRC grant EP/H009558.

5:02PM S26.00010 Temperature sensitivity of surface tension-driven flows: Application to time-temperature integration , JOHN THOMAS, LAWRENCE HUNTER, MICHAEL BOYLE, Johns Hopkins University Applied Physics Laboratory — The effects of time-dependent temperature fluctuations on surface-tension driven fluid flow inside a capillary are modeled using classical hydrodynamics. To begin, we use Newton's second law to derive a non-dimensional equation of motion that describes capillary flow as a function of system geometry, fluid properties, and fluid temperature. We use this model to examine how temperature excursions affect the instantaneous and long-term position and velocity of the fluid front inside the capillary. Next, we examine the combined effects of orientation change and temperature change on fluid movement through the capillary. Using this data, we show how to design a non-powered time-temperature integration device for recording the cumulative temperature exposure history of an asset or local environment. By selecting an appropriate fluid and capillary geometry, we show how such devices can be designed to exhibit arbitrary temperature sensitivities, operate over arbitrary monitoring periods (months to decades), and operate in a manner that does not depend on orientation.

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S27 Biofluids: General III Ballroom I

3:05PM S27.00001 Development of a Physiological Model for the Human Spine, MICHAEL KVITNITSKY, SIVA THANGAM, Stevens Institute of Technology — The intervertebral disc in a human spine is a complex structure consisting of three distinct parts: the nucleus pulposus, the annulus fibrosus, and the cartilaginous end-plates. The Nucleus Pulposus is centrally located within the disc surrounded by annulus fibrosus. It consists of a loose network of fibers and cells in a proteoglycan gel, which merges indistinctly at its outer margin with the annulus fibrosus. A viscoelastic constitutive model is proposed for the nucleus pulposus of the human spine to facilitate the development of a flexible intervetebral device designed for application in the thoraco-lumbar region of the human spine during surgery. A novel experimental set up was designed to establish application limits of the design concept for different approaches in spinal surgery. Both static and fatigue mechanical tests based on the ASTM standards provided a basis for the comparison with some existing clinically successful spinal implants designed for similar applications. Also, these mechanical tests and in-vitro comparison with normal spine provided the application limits of this design in surgery to maintain physiologic functional performance at the affected spinal level. The model is used to investigate the effect of the various design parameters on the biomechanical environment of the spine segment.

3:18PM S27.00002 Application of Control Volume Analysis to Cerebrospinal Fluid Dynamics, TIMOTHY WEI, University of Nebraska, BENJAMIN COHEN, RPI, TOMER ANOR, JOSEPH MADSEN, Harvard Medical School — Hydrocephalus is among the most common birth defects and may not be prevented nor cured. Afflicted individuals face serious issues, which at present are too complicated and not well enough understood to treat via systematic therapies. This talk outlines the framework and application of a control volume methodology to clinical Phase Contrast MRI data. Specifically, integral control volume analysis utilizes a fundamental, fluid dynamics methodology to quantify intracranial dynamics within a precise, direct, and physically meaningful framework. A chronically shunted, hydrocephalic patient in need of a revision procedure was used as an in vivo case study. Magnetic resonance velocity measurements within the patient's aqueduct were obtained in four biomedical state and were analyzed using the methods presented in this dissertation. Pressure force estimates were obtained, showing distinct differences in amplitude, phase, and waveform shape for different intracranial states within the same individual. Thoughts on the physiological and diagnostic research and development implications/opportunities will be presented.

3:31PM S27.00003 Lessons learned from the jellyfish: Fluid transport at intermediate Reynolds numbers, JANNA NAWROTH, JOHN DABIRI, Caltech — Biologically inspired hydrodynamic propulsion and maneuvering strategies promise the advancement of medical implants and novel robotic tools. We have chosen juvenile jellyfish as a model system for investigating fluid dynamics and morphological properties underlying fluid transport by an elastic system at intermediate Reynolds numbers. Recently we have described how natural variations in viscous forces are balanced by changes in jellyfish body shape (phenotypic plasticity), to the effect of facilitating efficient body-fluid interaction. Complementing these studies in our live model organisms, we are also engaged in engineering a synthetic jellyfish, that is, a rhythmically actuated elastomer capable of generating efficient feeding and propulsion currents. The main challenges here are (1) to derive a body shape and deformation suitable for effective fluid transport under physiological fluid conditions, (2) to understand the mechanical properties of actuator and elastomer to derive a design capable of the desired deformation, (3) to establish adequate 3D kinematics of power and recovery stroke, and (4) to evaluate the performance of the design.

3:44PM S27.00004 ABSTRACT WITHDRAWN –

3:57PM S27.00005 A flow separation study over a shortfin mako shark pectoral fin[1], MICHAEL BRADSHAW, AMY LANG, REDHA WAHIDI, DREW SMITH, The University of Alabama, PHILIP MOTTA, The University of South Florida — Many animals possess performance enhancing mechanisms, such as the denticles found on the skin of the shortfin mako shark (*Isurus oxyrinchus)*. The shortfin mako, one of the fastest sharks on the planet, is covered by small, tooth-like scales that vary in bristling capability. Previous biological findings have shown that the scales increase in flexibility from the leading to trailing edge over the pectoral fin. As this fin is a primary control surface, the scale bristling may provide a mechanism for separation control that leads to decreased drag and increased maneuverability. Such findings can potentially lead to the development of similar micro-scale mechanisms to improve the efficiency of aerospace design. A left pectoral fin (71 cm span) was tested in a water tunnel facility under static and dynamic conditions. Digital Particle Image Velocimetry (DPIV) was used to characterize the flow over the fin. Various angles of attack at two speeds were tested (Re of 44,500 and 68,000). Two chord-wise locations, approximately mid-span where three-dimensional effects were minimized, were viewed to analyze the flow. After the initial testing, the fin was painted to eliminate the effect of the scales and retested to observe flow separation.

[1]Supported by REU SITE EEC grant number 1062611

4:10PM S27.00006 A whisker sensor: role of geometry and boundary conditions, HENDRIK HANS, Nanyang Technological University, PABLO VALDIVIA Y ALVARADO, DILIP THEKOODAN, Singapore MIT Alliance for Research and Technology, MIAO JIANMIN, Nanyang Technological University, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology — Harbor seal whiskers are currently being studied for their role in sensing and tracking of the fluid structures left in wakes. Seal whiskers are exposed to incoming flows and are subject to self-induced vibrations. The whisker's unusual geometry is thought to reduce these self-induced disturbances and facilitate a stable reference for wake sensing. An experimental platform was designed to measure flow-induced displacements and vibrations at the base of whisker-like models. Four different whisker-like models (scale: 3x) were towed at different speeds down a towing tank and base displacements in the direction of motion and in the perpendicular axis were measured. Each model incorporated a particular geometrical feature found in harbor seal whiskers. Three different visco-elastic supports were used to mimic various boundary conditions at the base of the whisker models. The effects of geometrical features and boundary conditions on measured base vibrations at three relevant Reynolds numbers are discussed. The material properties of a model's base influence its sensitivity. When compared to a circular cylinder model, whisker models show almost no sign of VIV.

4:23PM S27.00007 Numerical Study of Seal Whisker Vibrations, GABRIEL WEYMOUTH, Singapore MIT Alliance for Research and Technology, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology — Harbor seal whiskers are thought to play an active role in the identification and tracking of wakes left by potential prey. Further, it is believed that the whisker's geometry enhances it's effectiveness as a sensor on the moving seal by minimizing self-induced fluid/structure excitations. In this study multiple test sections are simulated with an immersed-boundary numerical method at Re=500 to determine the geometry's influence on lift, drag, and wake structures in a fixed configuration. The results confirm findings that the whisker geometry is responsible for order of magnitude reductions in the lift force on the whisker and show that the whisker diminishes the strength of organized flow structures in the wake. Furthermore, prescribed, self-induced, and upstream-flow-induced vibration tests are preformed to asses if the fixed configuration results of the whisker morphology translate to increased effectiveness for wake detection.

4:36PM S27.00008 Harbor Seal Vibrissa Morphology Reduces Vortex-Induced Vibrations , HEATHER BEEM, JASON DAHL, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology — Studies show that harbor seals are adept at tracking small movements in the water, such as those left in the wake of fish, by using their highly sensitive whiskers to detect fluid structures, even without auditory or visual cues. The present work investigates the intriguing claim that the unique morphology of the harbor seal whisker suppresses Vortex Induced Vibrations (VIV) [1]. This implies that the geometry is specialized to reduce the background noise caused by the whisker's own wake in the detection of the upstream target. Forces on a rigid whisker model (scale: 50x) being towed steadily down a water tank while experiencing imposed oscillations are measured. A range of frequencies and amplitudes are tested, the hydrodynamic lift coefficient in phase with velocity ($C_{L,v}$) is calculated for each, and values are combined in a contour plot. The region of positive $C_{L,v}$ peaks at an amplitude ratio of 0.1, indicating that the whisker's undulatory, asymmetric structure considerably reduces (but does not entirely suppress) regions where the structure experiences VIV in comparison with a standard cylinder, whose peak reaches an amplitude ratio of 0.8.

[1] W. Hanke, et al., "Harbor seal vibrissa morphology suppresses vortex-induced vibrations," J. Exp. Biol., vol. 213, pp. 2665-72, 2010.

4:49PM S27.00009 Rectification of pulsatile stress on soft tissues: a mechanism for normal-pressure hydrocephalus , SHREYAS JALIKOP, SASCHA HILGENFELDT, The University of Illinois at Urbana-Champaign — Hydrocephalus is a pathological condition of the brain that occurs when cerebrospinal fluid (CSF) accumulates excessively in the brain cavities, resulting in compression of the brain parenchyma. Counter-intuitively, normal-pressure hydrocephalus (NPH) does not show elevated pressure differences across the compressed parenchyma. We investigate the effects of nonlinear tissue mechanics and periodic driving in this system. The latter is due to the cardiac cycle, which provides significant intracranial pressure and volume flow rate fluctuations. Nonlinear rectification of the periodic driving within a model of fluid flow in poroelastic material can lead to compression or expansion of the parenchyma, and this effect does not rely on changes in the mean intracranial pressure. The rectification effects can occur gradually over several days, in agreement with clinical studies of NPH.

5:02PM S27.00010 Modeling the population dynamics of phytoplankton in lacustrine ecosystems , TERRY JO LEITERMAN, St. Norbert College — Phytoplankton are microscopic plants, diverse in shape, and form the basis of aquatic ecosystems. Through both photosynthesis and respiration, they produce organic compounds and contribute notably to the Earth's carbon cycle, which make the population dynamics of phytoplankton important in discussions on climate change. In this talk, we introduce a model that predicts the vertical distribution of phytoplankton in freshwater lakes. The growth of phytoplankton is intimately connected to nutrient and light availability. Quantifying the growth due to light availability requires quantifying the seasonal settling velocity of the particles. Careful consideration is paid to the interaction between the forces of buoyancy, gravity, and drag. To accurately formulate settling velocity, the low Reynolds nature of the system is exploited and added to an experimental, laboratory component. The laboratory research is guided by the use of a sedimentation tank and a collection of vertical cylinders that allow the characterization of particle separation and settling velocity for sparse phytoplankton populations of both spherical and slender shape.

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S28 Wing Flexibility Ballroom II

3:05PM S28.00001 On passive wing response in flapping flight , ALBERT MEDINA, JEFF D. ELDREDGE, University of California, Los Angeles — The aerodynamic role of passive wing mechanics in biological flight remains poorly understood. This computational and experimental study focuses on the effects of flexibility on flapping foils. Our approach recognizes that deflections of insect wings primarily occur along major flexion lines, and therefore focuses on simplified wings with discrete rigid structures. In two-dimensional numerical simulations a wing consisting of rigid elliptical bodies connected by torsion springs is subjected to various acceleration profiles. Both the early-time behavior and sustained vortex shedding are investigated, and compared with an analogous rigid wing. It is found that the vortex shedding period is reduced with decreasing spring stiffness. These results are presented in the general context of vortex-induced vibrations. Our experiments focus on a dynamically scaled three-dimensional fruitfly wing in which we isolate deflection to flexion lines running largely spanwise or chordwise. We observe that previously-observed phenomena of the leading-edge vortex, such as vortex bursting, are affected by the passive response of the wing structure. Chordwise flexion has a significantly greater impact on this behaviour compared to spanwise flexion.

3:18PM S28.00002 Flexibility and inertia of flapping wings in forward flight[1] , FANG-BAO TIAN, HAOXIANG LUO, Vanderbilt University, XI-YUN LU, University of Science and Technology of China (USTC) — Insect wings typically deform passively in flight under the combined aerodynamic force and inertia of the wing. To study the effect of the wing flexibility on the aerodynamic performance, a two-dimensional numerical study is employed to simulate the fluid-structure interaction of an elastic plate performing forward flight. The leading edge of the plate is clamped, while the rest of the chord is free to deform, leading to passive pitching and a dynamic camber. The wing stiffness and mass ratio are varied, and their effects on the lift, thrust, and aerodynamic power are investigated. The results shows that the moderate chordwise deformation can improve both lift and thrust performance significantly. The instantaneous passive pitching angle and consequently the forces are largely affected by the mass ratio that determines whether the deformation is caused by the wing inertia or the aerodynamic force. The high mass ratio wings, whose deformation is due to the wing inertia, can produce more thrust than the low mass ratio wing at the same amount of deformation. However, the high thrust is gained at a price of more power requirement.

[1]This work is sponsored by the U.S. NSF and the NSF of China.

3:31PM S28.00003 Role of Wing/Body Flexibility in Insect Maneuver[1] , CHENGYU LI, HAIBO DONG, SAMANE ZEYGHAMI, Wright State University, FLOW SIMULATION RESEARCH GROUP(FSRG) TEAM — It's widely thought insects are able to accomplish fast maneuver via adjustment of wing kinematics. However, it's still unclear how wing flexibility plays roles in this process. In this work, an integrated study combining high-speed photogrammetry and direct numerical simulation (DNS), for a freely flying dragonfly (*Erythemis Simplicicollis*) in 110 degree turn, is used to reveal both aerodynamic and dynamic roles of its body and wings. Quantitative measurements have shown the significant difference of deformation between all wings as well as up to 18 degree bending of the tail. Unsteady 3D vortex formation and associated aerodynamic forces calculated from high-fidelity simulations are used to illustrate how the turn is accomplished within three dragonfly wing beats. This work is supported by NSF CBET-1055949.

[1]NSF CBET-1055949

3:44PM S28.00004 Understanding the Role of Chord-wise Flexibility in Flapping Wing Flight , ZACHARY GASTON, HAIBO DONG, HUI WAN, Wright State University, MICHAEL OL, Air Force Research Laboratory — Aerodynamic performance of flapping hinged plates is numerically studied to explore the effects of chord-wise flexibility in flapping wing flight. The plate with chord-wise flexibility is modeled as a two-link mechanism with a torsional spring hinge in between. The upper-link of the plate is controlled by prescribed motion and the rest of body is subjected to passive deflection due to fluid-body interaction. The effect of forced to natural frequency ratio is studied first for a flapping hinged-plate, on which prescribed hovering motion is actively applied. The effects of torsional stiffness and chord-wise flexibility are further explored for pitching and plunging plates, observing the flow phenomena and lift production as a result of this change. Comparisons between rigid plates, free-to-pivot hinged plates, and the torsional spring hinged plates are made, identifying a more optimal model for promoting lift production in flapping plates.

3:57PM S28.00005 Hovering performance of a two dimensional skeleton-reinforced flexible wing , KOUROSH SHOELE, Postdoctoral Research Associate, QIANG ZHU, University of California, San Diego — Skeleton-reinforced membrane is a typical biological design. An important application of these systems is in biolocomotion apparatus, most notably the wings of insects. The structural characteristics of the wing, a composite structure including a soft membrane reinforced by embedded skeleton(venation) is an important factor in performance of a insect flexible wing during hovering flight. To study how the structural anisotropy affects the aerodynamic performance of the deformable wing, a two-dimensional numerical study is applied to simulate the flow-structure interaction of a wing during hovering flight. In this two-dimensional rendition, the underlying veins are modeled as springs, and the membrane is modeled as a flexible plate. The effect of the wing anisotropy on lift production and power expenditure is studied for a range of veins rigidity, Reynolds number and wing inertia. It is shown that with flexible veins and the leading edge strengthening, the lift production can be significantly increased. In addition, the detailed distribution of veins stiffness in the wing has a significant effect on the unsteady flow pattern around the wing.

4:10PM S28.00006 Flow and forces of flexible wings under a plunging motion , DIEGO CAMPOS, LAWRENCE UKEILEY, University of Florida — The effects of flexibility on the flow fields and production of aerodynamic forces of Zimmerman planform wings with different stiffness are investigated. The wings are subjected to a symmetric sinusoidal plunging motion under forward flight conditions. Particle Image Velocimetry (PIV) is used to measure the flow at different spanwise locations and Laser Doppler Vibrometry (LDV) is utilized to obtain deformation characteristics from the wings. The results from the PIV and LDV analysis are phase averaged discrete points throughout the plunging cycle, resulting in three component flow field data coupled with wing twist obtained from the LDV, thus allowing for a better understanding of the fluid-structure interactions. The forces are then calculated through a momentum balance technique to better understand the effects of different stiffness. The Q criterion is used to identify and analyze the vortical structures that form around the wing. Results show the strongest leading edge vortex formation between 70 and 80 percent span. The effect of the deformation on the production and evolution of the vortical structures will be related to the generation of aerodynamic forces.

4:23PM S28.00007 Dynamics of Freely Swimming Flexible Foils , SILAS ALBEN, Georgia Tech, CHARLES WITT, T. VERNON BAKER, ERIK ANDERSON, Grove City College, GEORGE LAUDER, Harvard University — We use experiments, simulations, and modeling to study thin foils which are oscillated at the leading edge and are free to move unidirectionally under the resulting fluid forces. We find resonant-like peaks in the swimming speed as a function of foil length and rigidity. We find good agreement between the inviscid model and the experiment in the foil motions (particularly the wavelengths of their shapes), the dependences of their swimming speeds on foil length and rigidity, and the corresponding flows. The model predicts that the foil speed is proportional to foil length to the -1/3 power and foil rigidity to the 2/15 power. These scalings give a good collapse of the experimental data.

4:36PM S28.00008 Shape and Swimming of Flexible Pitching Foils , ANDERS ANDERSEN, BJARNE BACH, Department of Physics and Center for Fluid Dynamics, Technical University of Denmark, TEIS SCHNIPPER, Department of Mechanical Engineering and Center for Fluid Dynamics, Technical University of Denmark — We present an experimental study using flexible metal foils which are driven with simple harmonic pitching oscillations in water. We vary the frequency and amplitude of the oscillations and explore the foil response at frequencies both below and above the lowest resonance frequency. We compare our observations with theoretical predictions taking into account the elastic properties of the foil and using a simple model of the fluid forces. Finally, we make use of an experimental setup in which the pitching foils are mounted on a simple carousel that allows them to rotate (swim) freely and compare our observations on the rotation (swimming) velocity with that of rigid foils performing similar pitching oscillations.

4:49PM S28.00009 Bistable property of a flexible ring in a uniform flow[1] , BO YOUNG KIM, SOO JAI SHIN, HYUNG JIN SUNG, KAIST — An improved version of the immersed boundary (IB) method is developed for simulating a flexible ring clamped at one point in a uniform flow. The boundary of the ring consists of a flexible filament with tension and bending stiffness, which can be modeled as a linear spring with spring constant k and initially unstretched length. In our simulation, we observe bistable states, one stationary stable and the other self-sustained periodically flapping, that coexist over a range of flow velocities depending on the initial inclination angle. The bistable property of the initially elliptic flexible ring is observed in the simulations. The Reynolds number range of the bistability region and the flapping amplitude are specified for various aspect ratios (a/b). It is found that for a/b=0.5, the bistability region is most postponed and the flapping amplitude at the self-sustained flapping state is minimized. A new bistability phenomenon is observed that with certain aspect ratio two periodically flapping states coexist with different amplitudes in a Reynolds number range, instead of the stationary stable and periodically flapping states.

[1]This study was supported by the Creative Research Initiatives of NRF/MEST (No. 2011-0000423) of Korea.

5:02PM S28.00010 Scaling in Flexible Flapping Wings[1] , CHANG-KWON KANG, University of Michigan, HIKARU AONO[2], Japan Aerospace Exploration Agency, WEI SHYY[3], University of Michigan — The role of flexibility on the aerodynamic performance of a flapping wing is investigated. We consider chordwise, spanwise, and isotropic flexibility. Overall, the aerodynamic force is determined by the Reynolds number, reduced frequency (k), and Strouhal number (St). In particular, at the Reynolds number regime of O(10^3-10^4) and the reduced frequency of O(1), the added mass force, related to the acceleration of the wing, is important. Based on the order of magnitude and energy balance arguments, a relationship between the propulsive force and the maximum relative wing tip deformation parameter is established. The parameter depends on the density ratio, St, k, natural and flapping frequency ratio, and flapping amplitude. It seems that the maximum propulsive force is obtained when flapping near the resonance, whereas the optimal propulsive efficiency is reached when flapping at about half of the natural frequency; both are supported by the reported studies.

[1]This work was supported in part by the AFOSR's MURI grant and by the Michigan/AFRL Collaborative Center in Aeronautical Sciences.
[2]Previously: Postdoctoral scholar at the University of Michigan
[3]Also: Provost and Chair Professor at Hong Kong University of Science and Technology

Tuesday, November 22, 2011 3:05PM - 5:15PM –
Session S29 Swimming with Flagella Ballroom III

3:05PM S29.00001 Particle image velocimetry experiments on a model helical flagellum in viscoelastic fluids , ANAND DESAI, BIN LIU, THOMAS POWERS, KENNETH BREUER, School of Engineering, Brown University — Live bacteria often live in polymer suspensions, and are inevitably subjected to the effects of fluid viscoelasticity. To study the viscoelastic effect on bacterial motility, we have constructed a scaled-up model system and use particle image velocimetry (PIV) to measure the flow field generated by a rigid helical filament that rotates and translates in a Boger fluid. The helix is made to swim freely – it is subjected to an external torque, and translates along its axial direction at a predetermined speed so that the net hydrodynamic force on the helix vanishes. By comparing the flow field with the Newtonian reference, we address the question on how the viscoelasticity of the fluid enhances or reduces the motility of the helix at different Deborah numbers.

3:18PM S29.00002 Force-free swimming of a model helical flagellum in viscoelastic fluids , BIN LIU, THOMAS POWERS, KENNETH BREUER, School of Engineering, Brown University — Many bacteria swim by rotating helical flagella. These cells often live in polymer suspensions, which are viscoelastic. Recently there have been several theoretical and experimental studies showing that viscoelasticity can either enhance or suppress propulsion, depending on the details of the microswimmer. To help clarify this situation, we study experimentally the motility of the flagellum using a scaled-up model system - a motorized helical coil that rotates along its axial direction. The rotating helix is tethered on a linear stage that advances at prescribed speeds along the axial direction. A free-swimming speed is obtained when the net force on the helix is zero. In the Newtonian case, the free-swimming speed of the helix matches the predictions from the slender body theory and the boundary element method, with increasing order of agreement as the numerical strategy improves. When the helix is immersed in a viscoelastic (Boger) fluid, we find an increase in the force-free swimming speed. The enhancement is maximized at a Deborah number of approximately one, and the magnitude depends not only on the elasticity of the fluid but also on the geometry of the helix.

3:31PM S29.00003 Flagellar generated flow mediates attachment of *Giardia Lamblia* , THEODORE PICOU, JAMIE POLACKWICH, BEATRIZ BURROLA GABILONDO, RYAN MCALLISTER, Georgetown University, TOM POWERS, Brown University, HEIDI ELMENDORF, JEFF URBACH, Georgetown University — *Giardia lamblia* is a protozoan parasite responsible for widespread diarrheal disease in humans and animals worldwide. Attachment to the host intestinal mucosa and resistance to peristalsis is necessary for establishing infection, but the physical basis for this attachment is poorly understood. We report results from confocal fluorescence microscopy that demonstrate that the regular beating of the posterior flagella generate a flow through the ventral disk, a suction-cup shaped structure that is against the substrate during attachment. Finite element simulations show that the negative pressure generated by the flow is consistent with the measured force of attachement between the parasite and its substrate.

3:44PM S29.00004 Hydrodynamic effects on the tumbling of flagellated bacteria near a solid surface[1] , MEHDI MOLAEI, JIAN SHENG, Aerospace Engineering and Mechanics, University of Minnesota — Peritrichously flagellated bacteria use semi-rigid helical flagella to form a bundle during a run to swim forward and to trigger the unbundling during a tumble to change their swimming direction. It is accepted that the hydrodynamic interactions play a significant role in these processes. Recently, using digital holographic microscopy and microfluidics, we discovered that the tumbling events are substantially suppressed near a solid wall. In this paper, we present a two flagellum rigid model to elucidate the hydrodynamic wall interaction mechanism behind the phenomenon. Further implications on cell attachment and detachment during the biofilm formation will be discussed. We apply Slender Body Theory (SBT) to quantify the flow flagellum interaction. The no-slip boundary imposed by the wall is modeled using the image system of the SBT model for the stoke-flow singularity. We show that in the bulk, a repulsive force among flagella initiates the unbundling and consequently tumbling; however, in presence of the wall, the force is strongly mitigated that stabilize the bundle and suppress the tumbling.

[1]NIH and NSF

3:57PM S29.00005 Hydrodynamic Interactions of Hyperactivated Sperm , SARAH OLSON, Worcester Polytechnic Institute, LISA FAUCI, Tulane University — Hyperactivated sperm motility is correlated to an increase in calcium concentration within the flagellum and is characterized by highly asymmetrical waveforms and circular trajectories. Previous computational studies of flagellum with symmetrical waveforms have shown that multiple sperm swimming with waveforms that are out of phase will eventually phase lock due to hydrodynamic interactions. The focus of this research is to study the hydrodynamic interactions of hyperactivated sperm swimming in proximity in both an unbounded fluid domain and near a wall.

4:10PM S29.00006 Locomotive consequences of non-axisymmetric flagellar configurations , HENRY FU, University of Nevada, Reno, MARCOS MARCOS, Nanyang Technological University, YUNKYONG HYON, University of Nevada, Reno, THOMAS POWERS, Brown University, ROMAN STOCKER, Massachusetts Institute of Technology — Although peritrichous bacteria can form flagellar bundles at many attachment points and directions relative to the cell body, locomotion of these bacteria is often modeled as arising from a polar bundle oriented along the cell body axis. We discuss the consequences of non-axisymmetric flagellar configurations for bacterial locomotion and implications for bacterial behavior using a boundary element method (BEM) based on the method of regularized Stokeslets. We validate our BEM by comparing to analytic results for spheres and ellipsoids, as well as results in the literature for axisymmetric flagella with spherical and ellipsoidal heads obtained from other boundary element methods and slender body theory. Non-axisymmetric flagellar configurations generically lead to wobbling cell bodies and wiggling helical cell trajectories, both of which have been observed experimentally. We compare experimental and numerically calculated wiggling trajectories to deduce information about flagellar geometries of swimming B. subtilis. We discuss the implications of off-axis flagellar geometries for bacterial rheotaxis and chemotaxis.

4:23PM S29.00007 Propulsion of microorganisms by a helical flagellum , BRUCE RODENBORN, CHIH-HUNG CHEN, HARRY L. SWINNEY, H.P. ZHANG[1], University of Texas at Austin — Many microorganisms are propelled by rotating helical flagella. We examine this propulsion in laboratory measurements on macroscopic rotating helices (typical diameter, 12 mm) in a fluid with viscosity 10^5 times that of water; thus the Reynolds number in the experiments is much less than unity, just as for bacteria. We directly measure the propulsive force and torque generated by a rotating flagellum, and the drag force on a translating flagellum, i.e. elements of the propulsion matrices. Our results differ significantly from the predictions of Lighthill's Resistive Force Theory (1975), which treats each segment as an independent slender rod and neglects hydrodynamic interactions between segments of the flagellum. The difference between our measurements and Resistive Force Theory is especially large for helices with small pitch/diameter ratios, which is the regime of many bacteria. We also compute force, torque and drag using the regularized Stokelets method of Cortez et al. (2005). Our numerical results from the regularized Stokelets method are in excellent agreement with the laboratory measurements for helices with parameters (pitch/diameter and length/pitch) in the biologically relevant regime.

[1]Now at Shanghai Jiao Tong University

4:36PM S29.00008 Flagellar waveform analysis of swimming algal cells[1] , HUSEYIN KURTULDU, KARL JOHNSON, JERRY GOLLUB, Haverford College — The twin flagella of the green alga *Chlamydomas reinhardtii* are driven by dynein molecular motors to oscillate at about 50-60 Hz in a breaststroke motion. For decades, *Chlamydomas* has been used as a model organism for studies of flagellar motility, and of genetic disorders of ciliary motion. However, little is known experimentally about the flagellar waveforms, and the resulting time-dependent force distribution along the 250 nm diameter flagella. Here, we study flagellar dynamics experimentally by confining cells in quasi-2D liquid films. From simultaneous measurements of the cell body velocity and the time-dependent velocities along the center lines of the two flagella, we determine the drag coefficients, and estimate the power expended by the body and the flagella, comparing our findings with measurements[2] based on the induced fluid flow field. We contrast the results for the quite different beating patterns of synchronous and asynchronous flagella, respectively.

[1]Supported by NSF Grant DMR-0803153.

[2]J.S. Guasto et al., Phys. Rev. Lett. 105, 168102 (2010)

4:49PM S29.00009 Marine bacteria exploit Euler buckling to turn , KWANGMIN SON, JEFFREY S. GUASTO, ARNAUD LAZARUS, JAMES MILLER, PEDRO M. REIS, ROMAN STOCKER, MIT — Important species of marine bacteria were recently discovered to swim in a three-step pattern: they swim forward by rotating a single helical flagellum, then backwards by reversing the flagellar rotation, and finally "flick" the flagellum in an off-axis motion, producing a large ($\sim 90°$) reorientation in the swimming direction. What remains unknown in this elegant, minimalistic swimming pattern are the biomechanics of the flick. Here we present new observations based on high-speed video microscopy to capture the detailed dynamics of the reorientation process in *Vibrio alginolyticus*. Combining the data with a model of buckling of thin structures, we show that the onset of forward swimming triggers a mechanical instability of the flagellar hook, because the propulsive force exceeds the threshold for Euler buckling. This surprising adaptation, where cells take advantage of the flexibility of the flagellar hook to generate a turn, may represent the evolutionarily cheapest bacterial motility pattern and a highly beneficial solution to foraging in resource-poor marine environments.

5:02PM S29.00010 Investigation of the Swimming and Flagellar Dynamics of Shear guided Motile Alga[1] , ANWAR CHENGALA, MIKI HONDZO, Department of Civil Engineering, University of Minnesota, JIAN SHENG, Department of Aerospace Engineering and Mechanics, University of Minnesota — We examine the behavior of force-free swimming cell in a shear flow having spheroidal bodies with two flagella located at the anterior part of the cell that uses breast-like motion for its propulsion. The cell, *Dunaliella primolecta*, displays a unique behavior of propelling along the local vorticity direction in a linear shear flow. The cell rotation is absent during this display and the flagella beating is observed to be asynchronous. The forces and moments generated by the flagella are estimated numerically. Based on the Resistive Force Theory approach, we attempt to demonstrate that the moments generated by beating flagella and their alignment to flow are necessary to counter-act the vorticity of the flow. In addition, we explore the various mechanisms that cell may use to re-orient while in a shear flow as well as how critical the variations in the flagellar beating pattern are to a cell's swimming dynamics.

[1]NSF (No: CBET-0844647), Minnesota Futures Grant

Tuesday, November 22, 2011 3:05PM - 4:49PM –
Session S30 Biofluids: Respiration and Tear Films Ballroom IV

3:05PM S30.00001 Experiments on the fluid dynamics of the human cough , GARY SETTLES — Human coughing is studied non-intrusively by high-speed schlieren videography, revealing a turbulent jet lasting up to 1 sec with a total expelled air volume of about 2 L. Velocimetry of eddy motion reveals a jet centerline airspeed of at least 8 m/sec. With Re roughly 18,000 the cough jet is inertia-driven and buoyancy is negligible. It shows typical round-turbulent-jet behavior, including a conical spreading angle of 24 deg, despite irregular initial conditions. The cough jet is projected several m into the surrounding air before it mixes out. It is well known that a cough can transmit infectious agents, and we are advised to cover our mouths in an apparent attempt to thwart the jet formation. Present experiments have shown that wearing a surgical mask or respirator designed to prevent the inhalation of infectious agents also interferes with the cough-jet formation, redirecting it into the person's rising thermal plume. (Tang et al., J. Royal. Soc. Interface 6, S727, 2009.)

3:18PM S30.00002 ABSTRACT WITHDRAWN —

3:31PM S30.00003 Surface shear viscosity of a lung surfactant: Newtonian to non-Newtonian transition , AMIR SADOUGHI, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — DPPC molecule is the most prevalent constituent of lung surfactant, and understanding its behavior as a monolayer may lead to better simulations of respiration. At low surface pressures (i.e. large surface tensions, corresponding to area per molecule of about 50 angstrom squared, or greater), DPPC behaves as a purely viscous film with surface shear viscosity that is Reynolds number independent. Transition to a non-Newtonian regime occurs at large surface pressures. At the small scales associated with the liquid lining of the alveoli, the relative effects of surface viscosities can be comparable to that of surface tension. Here, we examine the interfacial hydrodynamics by isolating the effects of the surface shear viscosity. DPPC monolayer is spread from a concentrated solution at the air/water interface in a deep channel viscometer, consisting of an annular region between two stationary cylinders and a rotating floor. The interfacial velocity is measured non-invasively (without any seeding particles) using Brewster angle microscopy with short laser pulses. The departure from Newtonian behavior is quantified by comparisons to numerical simulations of Navier-Stokes with a Boussinesq–Scriven surface model and various surface shear viscosities.

3:44PM S30.00004 The influence of surfactant on the propagation of a semi-infinite bubble through a liquid- filled compliant channel[1] , DAVID HALPERN, University of Alabama, DONALD GAVER, Tulane University — Pulmonary airway closure may occur as a result of fluid accumulation and surfactant insufficiency. This results in "compliant collapse" with the airway walls buckled and held in apposition by a liquid plug that blocks the passage of air. Airway reopening is vital to the recovery of adequate ventilation, but has been associated with ventilator- induced injury because of the exposure of airway epithelial cells to large pressure gradients. Surfactant replacement is helpful in modulating this harmful stimulus, but is limited in its effectiveness due to slow surfactant adsorption. We investigate the effect of surfactant on reopening by considering the steady two-dimensional motion of a semi-infinite bubble propagating through a liquid-filled compliant channel doped with soluble surfactant. Many parameters affect reopening, but we primarily investigate the capillary number Ca (the ratio of viscous to surface tension forces), the reopening (bubble) pressure p_b, the adsorption depth parameter λ (a bulk concentration parameter) and the bulk Peclet number Pe_b (the ratio of bulk convection to diffusion). The behavior of this system, and the impact of the flow field on surfactant transport and the applied stresses on the wall will be discussed.

[1]Funded by NIH R01-HL81266.

3:57PM S30.00005 Tear film dynamics with evaporation and osmolarity , JAVED SIDDIQUE, Penn State, York, RICHARD BRAUN, University of Delaware — We consider a model problem for the evaporation and breakup up of tear film. The model includes the effects of surface tension, Marangoni stresses, insoluble surfactant transport, evaporation, osmolarity transport, osmosis and wetting of corneal surface. Evaporation is made dependent on surface concentration in order to mimic the lipid layer of the tear film when there is a single fluid layer in the model. In many cases for a single layer, the Marangoni effect seems to eliminate a localized area of increased evaporation due to reduced surfactant concentration. In this model the osmolarity in the tear film increases because of average evaporation rate rather than by a locally increased evaporation rate. If time permits, the effect of having a second fluid layer, representing the lipid layer, will be explored as well.

4:10PM S30.00006 Modeling the evaporation of a pre-lens tear film on a contact lens[1] ,

DANIEL ANDERSON, KEVIN TALBOTT, George Mason University, AMBER XU, Carnegie Mellon University, PADMANABHAN SESHAIYER, George Mason University — We develop a model for evaporation of a post-blink pre-lens tear film in the presence of a porous contact lens. The tear film is modeled as a Newtonian fluid and the flow in the contact lens is assumed to obey Darcy's law. The evaporation model treats the contact lens as a wetting surface. Evaporative mass flux thins the pre-lens film down to a nonzero steady thickness at which point evaporation continues by drawing fluid up through the contact lens. Both one and two-dimensional models are explored. The post-lens film (between the contact lens and the corneal surface) is not included explicitly in the model but is assumed to act as a reservoir that supplies fluid drawn up through the contact lens.

[1]Supported by NSF CSUMS and REU programs.

4:23PM S30.00007 A Two-Dimensional Model for Tear Flow through a Permeable Contact Lens[1] ,

MATTHEW GERHART, George Mason University, DANIEL ANDERSON, George Mason Universty — This is an investigation into the flow of an incompressible fluid in a thin pre-lens, contact lens, and post-lens system. The pre-lens and post-lens layers are coupled through the middle layer being a thin permeable contact lens. Included in this system are the effects of evaporation of the pre-lens film and the settling of the contact lens over time. This analysis is done through the nondimensionalization of the governing equations which include the Navier-Stokes' equations for the pre- and post-lens tear film and Darcy's equations for the contact lens layer along with heat equations for temperature. Lubrication theory is then applied to the resulting system leaving a reduced coupled system of equations whose properties we explore.

[1]Supported by NSF-DMS.

4:36PM S30.00008 Tear Film Dynamics and Cooling of the Anterior Eye ,

LONGFEI LI, RICHARD BRAUN, University of Delaware — A model for cooling of the human tear film is formulated by incorporating the heat transfer from the interior of the eye. A single PDE that governs the thickness of the film is derived from lubrication theory; the nonlinear partial differential equation is to be solved along the edge of a model rectangular domain for the anterior eye that includes the cornea (thin and thick substrate cases are treated separately). Optimal parameters to reproduce observed temperature decreases are found.